Computer Numerical Control Products

Series 20-TA

Operator's Manual

GFZ-62204E/02 *December 1994*

In this manual we have tried as much as possible to describe all the various matters.
However, we cannot describe all the matters which must not be done, or which cannot be done, because there are so many possibilities.
Therefore, matters which are not especially described as possible in this manual should be regarded as "impossible".

Warnings, Cautions, and Notes as Used in this Publication

Warning

Warning notices are used in this publication to emphasize that hazardous voltages, currents, temperatures, or other conditions that could cause personal injury exist in this equipment or may be associated with its use.

In situations where inattention could cause either personal injury or damage to equipment, a Warning notice is used.

Caution

Caution notices are used where equipment might be damaged if care is not taken.

Note

Notes merely call attention to information that is especially significant to understanding and operating the equipment.

This document is based on information available at the time of its publication. While efforts have been made to be accurate, the information contained herein does not purport to cover all details or variations in hardware or software, nor to provide for every possible contingency in connection with installation, operation, or maintenance. Features may be described herein which are not present in all hardware and software systems. GE Fanuc Automation assumes no obligation of notice to holders of this document with respect to changes subsequently made.

GE Fanuc Automation makes no representation or warranty, expressed, implied, or statutory with respect to, and assumes no responsibility for the accuracy, completeness, sufficiency, or usefulness of the information contained herein. No warranties of merchantability or fitness for purpose shall apply.

©Copyright 1995 GE Fanuc Automation North America, Inc.
All Rights Reserved

GE Fanuc: Your Source For The Best CNC

To locate your Sales Engineers and to get Literature and Brochures call:
USA: **1-800-648-2001** § Canada: **1-800-668-4640**

Customer Service:

- **Order placement**
- **Order status**
- **Parts price & delivery**
- **Shipping information**

Machine Tool Builders: 1-800-432-2621
FAX: 1-800-432-3291
All others: 1-800-828-6151
FAX: 1-804-978-5516

The Technical Support Center (800)828-6151

GE Fanuc has established the Technical Support Center to assist you with technical problems and questions. The Technical Support Center is available twenty-four hours a day, 365 days a year.

Part Programming Systems (708)932-6106

GE Fanuc offers a range of devices for CNC part program management. A full range of part programming software is also available.

Field Service (800)828-6151

Should you have a more complex problem, GE Fanuc will dispatch one of its service engineers to your location. We have field service offices in California, Connecticut, Illinois, Indiana, Michigan, New York, North Carolina, Ohio, Texas, Virginia, Ontario, Quebec (Canada) and Nuevo Leon (Mexico).

Training (800)828-6151

We have designed a special training facility for our customers in Charlottesville, Virginia and maintain facilities in Chicago and Detroit. All of our CNC training programs feature experienced instructors, quality documentation, and practical hands-on lab exercises.

Service Contracts (800)828-6151

GE Fanuc offers a variety of service contracts, which may be tailored to a customer's specific needs. Spot service is available on an hourly basis. Another service for our customers is the GE Fanuc Start-Up Assistance Program.

Documentation

Information on GE Fanuc controls is available in a wide range of instruction books. These include description manuals, operator's manuals, connection manuals, programming manuals, and maintenance manuals. Contact numbers listed above.

Parts and Parts Repair Service

GE Fanuc maintains an extensive parts inventory at our Charlottesville headquarters for all our CNCs. This inventory is outlined in our free Renewal Parts Catalog available upon request. We also provide a repair and return service for our customers. Contact numbers listed above.

Warranty Support (800)828-6151

GE Fanuc products have exceptional reliability. However, should a failure occur within the warranty period, our Technical Support Center will assist you to replace the failed item and if you wish, will dispatch a field engineer to troubleshoot the problem.

CNC Express (800)327-8262

CNC Express provides easy ordering and quick delivery of the world's best CNC products in standard configurations. You can order your CNC or Motion Control systems direct from our factory, with delivery from stock in as little as 4 weeks, using our CNC Express Catalog.

Options and Enhancements (804)978-5545

With GE Fanuc options, you can upgrade your CNC technology without scrapping your existing machine tools. If you need more tool offsets, probe and gauging capability, larger part program storage, improved programming functions, GE Fanuc can do the job.

GE FANUC

GE Fanuc Automation North America, Inc.
P.O. Box 8106 - CNC Business
Charlottesville, VA 22906

All specifications are subject to change without notice. No part of this document may be reproduced in any form.

Table of Contents

I. GENERAL

1. GENERAL 3

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL 5
1.2 NOTES ON READING THIS MANUAL 7

II. PROGRAMMING

1. GENERAL 11

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE–INTERPOLATION 12
1.2 FEED–FEED FUNCTION 15
1.3 PART DRAWING AND TOOL MOVEMENT 16
1.3.1 Reference Position (Machine–Specific Position) 16
1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System 17
1.3.3 How to Indicate Command Dimensions for Moving the Tool–Absolute, Incremental Commands 20
1.4 CUTTING SPEED–SPINDLE SPEED FUNCTION 23
1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING–TOOL FUNCTION 24
1.6 COMMAND FOR MACHINE OPERATIONS–MISCELLANEOUS FUNCTION 25
1.7 PROGRAM CONFIGURATION 26
1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM 29
1.9 TOOL MOVEMENT RANGE–STROKE 30

2. CONTROLLED AXES 31

2.1 CONTROLLED AXES 32
2.2 NAMES OF AXES 32
2.3 INCREMENT SYSTEM 33
2.4 MAXIMUM STROKES 34

3. PREPARATORY FUNCTION (G FUNCTION) 35

4. INTERPOLATION FUNCTIONS 38

4.1 POSITIONING (G00) 39
4.2 LINEAR INTERPOLATION (G01) 41
4.3 CIRCULAR INTERPOLATION (G02,G03) 42
4.4 CONSTANT LEAD THREADING (G32) 46
4.5 MULTIPLE–THREAD CUTTING 50
4.6 SKIP FUNCTION (G31) 51

5. FEED FUNCTIONS 53

5.1 GENERAL 54
5.2 RAPID TRAVERSE 55
5.3 CUTTING FEED 56

5.4 DWELL (G04) 59
5.5 AUTOMATIC ACCELERATION/DECELERATION 60
5.5.1 Automatic Acceleration/Deceleration 60
5.5.2 Linear Acceleration/ Deceleration after Interpolation for Cutting Feed 62

6. REFERENCE POSITION 64

7. COORDINATE SYSTEM 67

7.1 MACHINE COORDINATE SYSTEM 68
7.2 WORKPIECE COORDINATE SYSTEM 69
7.2.1 Setting a Workpiece Coordinate System (Method Using G50) 69
7.2.2 Automatic Coordinate System Setting 71
7.2.3 Workpiece Coordinate System Shift 72

8. COORDINATE VALUE AND DIMENSION 73

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91) 74
8.2 INCH/METRIC CONVERSION (G20, G21) 75
8.3 DECIMAL POINT PROGRAMMING 76
8.4 DIAMETER AND RADIUS PROGRAMMING 78

9. SPINDLE SPEED FUNCTION 79

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE 80
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND) 80
9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97) 80

10. TOOL FUNCTION (T FUNCTION) 84

10.1 TOOL SELECTION 85

11. AUXILIARY FUNCTION 86

11.1 AUXILIARY FUNCTION (M FUNCTION) 87
11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK 88

12. PROGRAM CONFIGURATION 89

12.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS 91
12.2 PROGRAM SECTION CONFIGURATION 94
12.3 SUBPROGRAM 100

13. FUNCTIONS TO SIMPLIFY PROGRAMMING 104

13.1 CANNED CYCLE (G90, G92, G94) 105
13.1.1 Outer Diameter/Internal Diameter Cutting Cycle (G90) 105
13.1.2 Thread Cutting Cycle (G92) 107
13.1.3 End Face Turning Cycle (G94) 109
13.1.4 How to Use Canned Cycles (G90, G92, G94) 112
13.2 CHAMFERING AND CORNER R 114

14. COMPENSATION FUNCTION 117

14.1 TOOL OFFSET 118

14.1.1 Tool Geometry Offset and Tool Wear Offset ... 118
14.1.2 T Code for Tool Offset ... 119
14.1.3 Tool Selection ... 119
14.1.4 Offset Number ... 119
14.1.5 Offset ... 120
14.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION ... 123
14.2.1 Imaginary Tool Nose ... 123
14.2.2 Direction of Imaginary Tool Nose ... 125
14.2.3 Offset Number and Offset Value ... 126
14.2.4 Work Position and Move Command ... 128
14.2.5 Notes on Tool Nose Radius Compensation ... 133
14.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION ... 136
14.3.1 General ... 136
14.3.2 Tool Movement in Start–up ... 138
14.3.3 Tool Movement in Offset Mode ... 140
14.3.4 Tool Movement in Offset Mode Cancel ... 153
14.3.5 Interference Check ... 156
14.3.6 Overcutting by Tool Nose Radius Compensation ... 161
14.3.7 Correction in Chamfering and Corner Arcs ... 163
14.3.8 Input Command from MDI ... 165
14.3.9 General Precautions for Offset Operations ... 166
14.4 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10) ... 167
14.4.1 Tool Compensation and Number of Tool Compensation ... 167
14.4.2 Changing of Tool Offset Value (Programmable Data Input) (G10) ... 168

15. CUSTOM MACRO ... 169

15.1 VARIABLES ... 170
15.2 SYSTEM VARIABLES ... 174
15.3 ARITHMETIC AND LOGIC OPERATION ... 180
15.4 MACRO STATEMENTS AND NC STATEMENTS ... 184
15.5 BRANCH AND REPETITION ... 185
15.5.1 Unconditional Branch (GOTO Statement) ... 185
15.5.2 Conditional Branch (IF Statement) ... 185
15.5.3 Repetition (WHILE Statement) ... 186
15.6 MACRO CALL ... 189
15.6.1 Simple Call (G65) ... 189
15.6.2 Modal Call (G66) ... 194
15.6.3 Macro Call Using G Code ... 196
15.6.4 Macro Call Using an M Code ... 197
15.6.5 Subprogram Call Using an M Code ... 198
15.6.6 Subprogram Calls Using a T Code ... 199
15.6.7 Sample Program ... 200
15.7 PROCESSING MACRO STATEMENTS ... 202
15.8 REGISTERING CUSTOM MACRO PROGRAMS ... 204
15.9 LIMITATIONS ... 205
15.10 EXTERNAL OUTPUT COMMANDS ... 206

16. PROGRAMMABLE PARAMETER ENTRY (G10) 210

III. OPERATION

1. GENERAL 215

1.1 MANUAL OPERATION 216
1.2 TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION 218
1.3 AUTOMATIC OPERATION 219
1.4 TESTING A PROGRAM 221
1.4.1 Check by Running the Machine 221
1.4.2 How to View the Position Display Change without Running the Machine 222
1.5 EDITING A PART PROGRAM 223
1.6 DISPLAYING AND SETTING DATA 224
1.7 DISPLAY 227
1.7.1 Program Display 227
1.7.2 Current Position Display 228
1.7.3 Alarm Display 228
1.7.4 Parts Count Display, Run Time Display 229
1.8 DATA INPUT/OUTPUT 230

2. OPERATIONAL DEVICES 231

2.1 CRT/MDI PANELS AND LCD/MDI PANELS 232
2.2 FUNCTION KEYS AND SOFT KEYS 235
2.2.1 General Screen Operations 235
2.2.2 Function Keys 236
2.2.3 Soft Keys 237
2.2.4 Key Input and Input Buffer 253
2.2.5 Warning Messages 254
2.2.6 Function Selection Soft Keys 255
2.3 EXTERNAL I/O DEVICES 256
2.3.1 FANUC Handy File 258
2.3.2 FANUC Floppy Cassette 258
2.3.3 FANUC FA Card 259
2.3.4 FANUC PPR 259
2.3.5 Portable Tape Reader 260
2.4 POWER ON/OFF 261
2.4.1 Turning on the Power 261
2.4.2 Screen Displayed at Power–on 262
2.4.3 Power Disconnection 263

3. MANUAL OPERATION 264

3.1 MANUAL REFERENCE POSITION RETURN 265
3.2 JOG FEED 267
3.3 INCREMENTAL FEED 269
3.4 MANUAL HANDLE FEED 270

3.5 MANUAL ABSOLUTE ON AND OFF 272

4. AUTOMATIC OPERATION 277

4.1 MEMORY OPERATION 278
4.2 MDI OPERATION (GUIDANCE PROGRAMMING MDI OPERATION) 281
4.3 DNC OPERATION 288
4.4 MANUAL HANDLE INTERRUPTION 289
4.5 MIRROR IMAGE 292
4.6 MANUAL INTERVENTION AND RETURN 294

5. TEST OPERATION 296

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK 297
5.2 FEEDRATE OVERRIDE 299
5.3 RAPID TRAVERSE OVERRIDE 300
5.4 DRY RUN 301
5.5 SINGLE BLOCK 303

6. SAFETY FUNCTIONS 305

6.1 EMERGENCY STOP 306
6.2 OVERTRAVEL 307
6.3 STROKE CHECK 308

7. ALARM AND SELF–DIAGNOSIS FUNCTIONS 311

7.1 ALARM DISPLAY 312
7.2 ALARM HISTORY DISPLAY 314
7.3 CHECKING BY SELF–DIAGNOSTIC SCREEN 315

8. DATA INPUT/OUTPUT 318

8.1 FILES 319
8.2 FILE SEARCH 321
8.3 FILE DELETION 323
8.4 PROGRAM INPUT/OUTPUT 324
8.4.1 Inputting a Program 324
8.4.2 Outputting a Program 326
8.5 OFFSET DATA INPUT AND OUTPUT 328
8.5.1 Inputting Offset Data 328
8.5.2 Outputting Offset Data 329
8.6 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA 330
8.6.1 Inputting Parameters 330
8.6.2 Outputting Parameters 331
8.6.3 Inputting Pitch Error Compensation Data 332
8.6.4 Outputting Pitch Error Compensation Data 333
8.7 INPUTTING/ OUTPUTTING CUSTOM MACRO COMMON VARIABLES 334
8.7.1 Inputting Custom Macro Common Variables 334
8.7.2 Outputting Custom Macro Common Variable 335

9. EDITING PROGRAMS 336

9.1 EDITING WITH GUIDANCE PROGRAMMING 338
9.1.1 Displaying the Contents of a Program 338
9.1.2 Modifying the Contents of a Program 340
9.1.3 Deleting a Block 341
9.1.4 Inserting a Block 342
9.1.5 Deleting a Program 343
9.1.6 Copying a Program 344
9.1.7 Renaming a Program 345
9.1.8 Editing in CNC Language 347
9.2 INSERTING, ALTERING AND DELETING A WORD 349
9.2.1 Word Search 350
9.2.2 Heading a Program 352
9.2.3 Inserting a Word 353
9.2.4 Altering a Word 354
9.2.5 Deleting a Word 355
9.3 DELETING BLOCKS 356
9.3.1 Deleting a Block 356
9.3.2 Deleting Multiple Blocks 357
9.4 PROGRAM NUMBER SEARCH 358
9.5 SEQUENCE NUMBER SEARCH 359
9.6 DELETING PROGRAMS 361
9.6.1 Deleting One Program 361
9.6.2 Deleting All Programs 361
9.6.3 Deleting More Than One Program by Specifying a Range 362
9.7 EDITING OF CUSTOM MACROS 363
9.8 BACKGROUND EDITING 364
9.9 PASSWORD FUNCTION 366

10. CREATING PROGRAMS 368

10.1 CREATING A PROGRAM WITH GUIDANCE PROGRAMMING 369
10.1.1 Program Menu Screen 369
10.1.2 Registering a Program 371
10.1.3 Comment 374
10.1.4 Guidance Programming 375
10.1.5 Warning Messages in Guidance Programming 395
10.2 CREATING A PROGRAM IN THE CNC LANGUAGE ON THE MDI PANEL 396
10.3 AUTOMATIC INSERTION OF SEQUENCE NUMBERS 398
10.4 CREATING PROGRAMS IN TEACH IN MODE 400

11. SETTING AND DISPLAYING DATA 403

11.1 SCREENS DISPLAYED BY FUNCTION KEY 412
11.1.1 Position Display in the Workpiece Coordinate System 413
11.1.2 Position Display in the Relative Coordinate System 414
11.1.3 Overall Position Display 416
11.1.4 Actual Feedrate Display 417

11.1.5 Display of Run Time and Parts Count 419
11.1.6 Operating Monitor Display 420
11.2 SCREENS DISPLAYED BY FUNCTION KEY [PROG] (IN MEMORY MODE OR MDI MODE) 421
11.2.1 Program Contents Display 422
11.2.2 Current Block Display Screen 423
11.2.3 Next Block Display Screen 424
11.2.4 Program Check Screen 425
11.2.5 Program Screen for MDI Operation 426
11.3 SCREENS DISPLAYED BY FUNCTION KEY [PROG] (IN THE EDIT MODE) 427
11.3.1 Displaying Memory Used and a List of Programs 427
11.4 SCREENS DISPLAYED BY FUNCTION KEY [OFFSET SETTING] 430
11.4.1 Displaying and Setting the Tool Offset Value 431
11.4.2 Direct Input of Tool Offset Value 434
11.4.3 Counter Input of Offset Value 436
11.4.4 Setting the Workpiece Coordinate System Shifting Amount 437
11.4.5 Displaying and Entering Setting Data 439
11.4.6 Displaying and Setting Run Time,Parts Count, and Time 441
11.4.7 Displaying and Setting Custom Macro Common Variables 443
11.4.8 Displaying and Setting the Software Operator's Panel 444
11.5 SCREENS DISPLAYED BY FUNCTION KEY [SYSTEM] 446
11.5.1 Displaying and Setting Parameters 447
11.5.2 Displaying and Setting Pitch Error Compensation Data 449
11.6 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION 451
11.6.1 Displaying the Program Number and Sequence Number 451
11.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation 452

12. HELP FUNCTION 454

IV. MAINTENANCE

1. METHOD OF REPLACING BATTERY 461

1.1 REPLACING CNC BATTERY FOR MEMORY BACK–UP 462
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER 464
1.3 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER (SERVO AMPLIFIER CONVERTER UNIT ONLY FOR SERIES 20 OR α SERIES SERVO AMPLIFIER MODULE) 465

APPENDIX

A.TAPE CODE LIST 469

B.LIST OF FUNCTIONS AND TAPE FORMATC 472

C.RANGE OF COMMAND VALUE 475

D.NOMOGRAPHS 477

D.1 INCORRECT THREADED LENGTH 478

D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH 480
D.3 TOOL PATH AT CORNER 482
D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING 485

E. STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET 486

F. CHARACTER–TO–CODES CORRESPONDENCE TABLE 488

G. ALARM LIST 489

H. OPERATION OF PORTABLE TAPE READER 503

I. GENERAL

1 GENERAL

About this manual

This manual consists of the following parts:

I. GENERAL
Describes chapter organization, applicable models, related manuals, and notes for reading this manual.

II. PROGRAMMING
Describes each function: Format used to program functions in the NC language, characteristics, and restrictions.

III. OPERATION
Describes the manual operation and automatic operation of a machine, procedures for inputting and outputting data, and procedures for editing a program.
Refer to the Machining Guidance Function OPERATOR'S MANUAL (B–62204E–1) for details of the machining guidance function.

IV. MAINTENANCE
Describes alarms, self–diagnosis, and procedures for replacing fuses and batteries.

V. APPENDIX
Lists tape codes, valid data ranges, and error codes.

This manual does not describe parameters in detail. For details on parameters mentioned in this manual, refer to the manual for parameters (B–62210E).

This manual describes all optional functions. Look up the options incorporated into your system in the manual written by the machine tool builder.
The models covered by this manual, and their abbreviations are:

Applicable models

Product name	Abbreviations	
FANUC Series 20–TA	20–TA	Series 20

Special symbols

This manual uses the following symbols:

IP_ : Indicates a combination of axes such as X_ Z_ (used in PROGRAMMING.).

; : Indicates the end of a block. It actually corresponds to the ISO code LF or EIA code CR.

Related manuals

The table below lists manuals related to the FANUC Series 20–TA. In the table, this manual is marked with an asterisk (*).

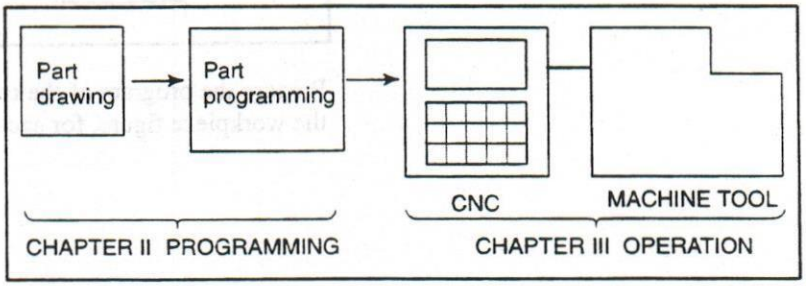
Table 1 Manuals Related to the FANUC Series 20–TA

Manual name	Specification number	
FANUC Series 20 DESCRIPTIONS	B–62172E	
FANUC Series 20 CONNECTION MANUAL	B–62173E	
FANUC Series 20–TA OPERATOR'S MANUAL	B–62204E	*
FANUC Series 20–TA MACHINING GUIDANCE FUNCTION OPERATOR'S MANUAL	B–62204E–1	
FANUC Series 20 MAINTENANCE MANUAL	B–62175E	
FANUC Series 20–TA PARAMETER MANUAL	B–62210E	
FANUC Series 16/18/20 PROGRAMMING MANUAL (Macro Compiler / Macro executer)	B–61803E–1	
FANUC MACRO COMPILER PROGRAMMING MANUAL	B–66102E	

1.1 GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL

When machining the part using the CNC machine tool, first prepare the program, then operate the CNC machine by using the program.

1) First, prepare the program from a part drawing to operate the CNC machine tool.
   How to prepare the program is described in the Chapter II. PROGRAMMING.
2) The program is to be read into the CNC system. Then, mount the workpieces and tools on the machine, and operate the tools according to the programming. Finally, execute the machining actually.
   How to operate the CNC system is described in the Chapter III. OPERATION.

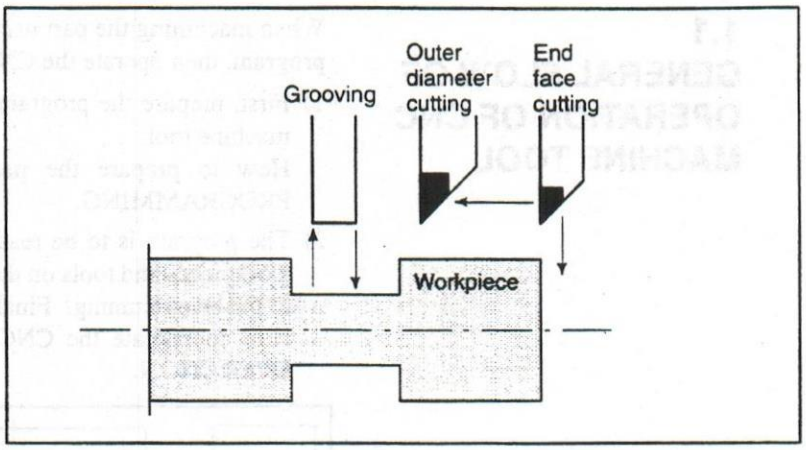


Before the actual programming, make the machining plan for how to machine the part.
Machining plan

1. Determination of workpieces machining range
2. Method of mounting workpieces on the machine tool
3. Machining sequence in every cutting process
4. Cutting tools and cutting conditions

Decide the cutting method in every cutting process.

Cutting process / Cutting procedure	1 End face cutting	2 Outer diameter cutting	3 Grooving
1. Cutting method : Rough Semi Finish			
2. Cutting tools			
3. Cutting conditions : Feedrate Cutting depth			
4. Tool path			



Prepare the program of the tool path and cutting condition according to the workpiece figure, for each cutting.

1.2 NOTES ON READING THIS MANUAL

1) The function of an CNC machine tool system depends not only on the CNC, but on the combination of the machine tool, its magnetic cabinet, the servo system, the CNC, the operator's panels, etc. It is too difficult to describe the function, programming, and operation relating to all combinations. This manual generally describes these from the stand–point of the CNC. So, for details on a particular CNC machine tool, refer to the manual issued by the machine tool builder, which should take precedence over this manual.
2) Headings are placed in the left margin so that the reader can easily access necessary information. When locating the necessary information, the reader can save time by searching though these headings.

Machining programs, parameters, variables, etc. are stored in the CNC unit internal non–volatile memory. In general, these contents are not lost by the switching ON/OFF of the power. However, it is possible that a state can occur where precious data stored in the non–volatile memory has to be deleted, because of deletions from a maloperation, or by a failure restoration. In order to restore rapidly when this kind of mishap occurs, it is recommended that you create a copy of the various kinds of data beforehand.

This manual describes as many reasonable variations in equipment usage as possible. It cannot address every combination of features, options and commands that should not be attempted.
If a particular combination of operations is not described, it should not be attempted.

II. PROGRAMMING

1 GENERAL

1.1 TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE– INTERPOLATION

Explanations

- **Tool movement along a straight line**

The tool moves along straight lines and arcs constituting the workpiece parts figure (See II–4).

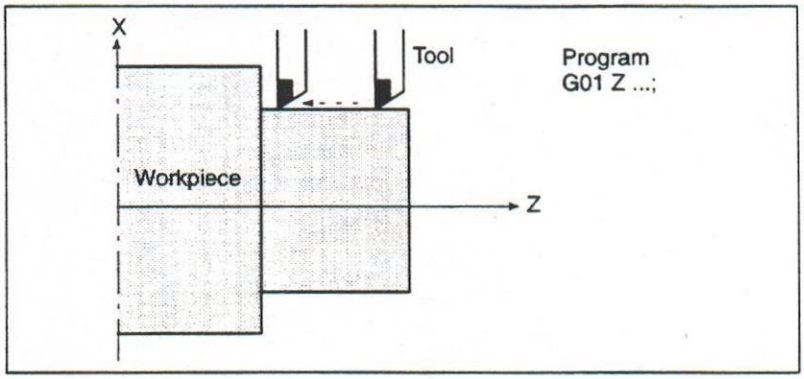


Fig.1.1 (a) Tool movement along the straight line which is parallel to Z–axis

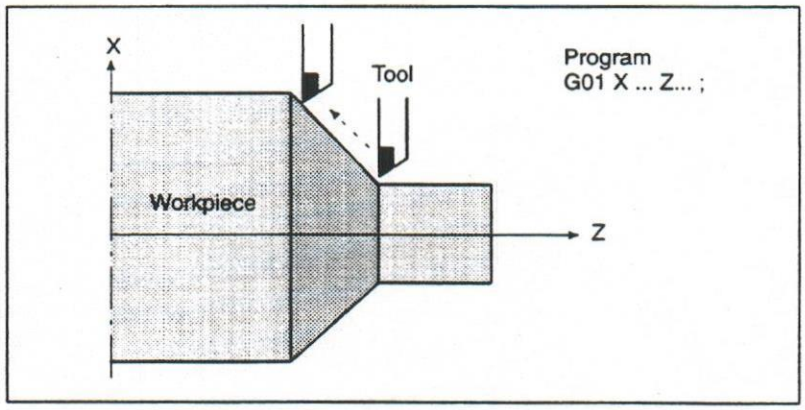


Fig.1.1 (b) Tool movement along the taper line

- **Tool movement along an arc**

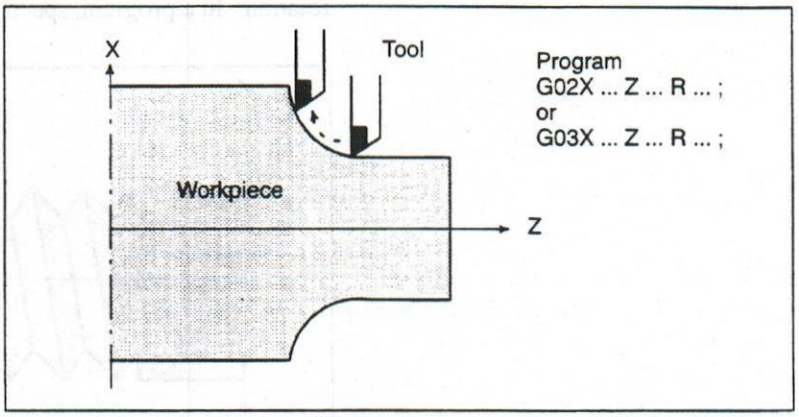


Fig. 1.1 (c) Tool movement along an arc

The term interpolation refers to an operation in which the tool moves along a straight line or arc in the way described above.
Symbols of the programmed commands G01, G02, ... are called the preparatory function and specify the type of interpolation conducted in the control unit.

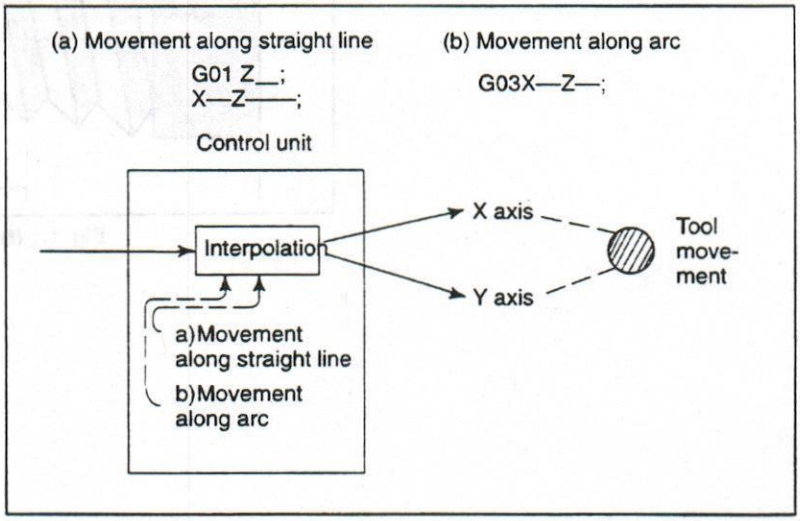


Fig. 1.1 (d) Interpolation function

Notes
Some machines move tables instead of tools but this manual assumes that tools are moved against workpieces.

- **Thread cutting**

Threads can be cut by moving the tool in synchronization with spindle rotation. In a program, specify the thread cutting function by G32.

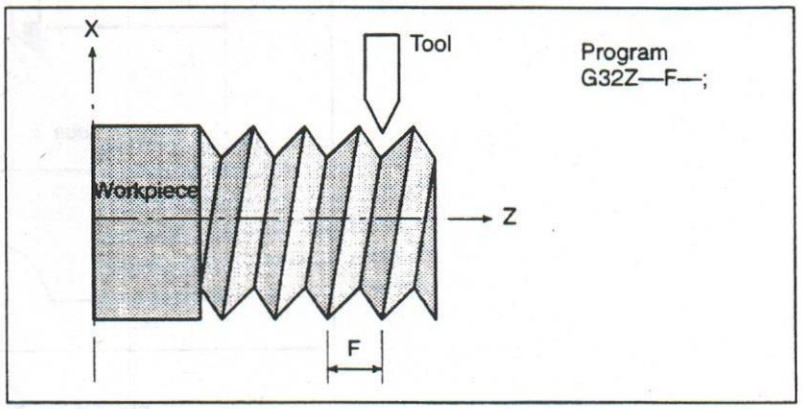


Fig. 1.1 (e) Straight thread cutting

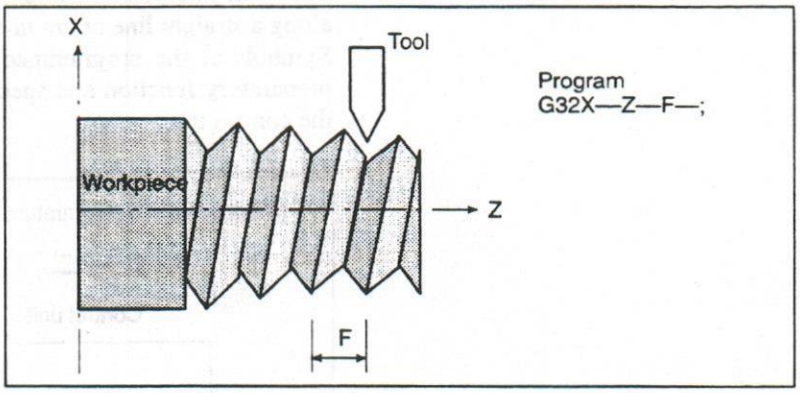


Fig. 1.1 (f) Taper thread cutting

1.2 FEED–FEED FUNCTION

Movement of the tool at a specified speed for cutting a workpiece is called the feed.

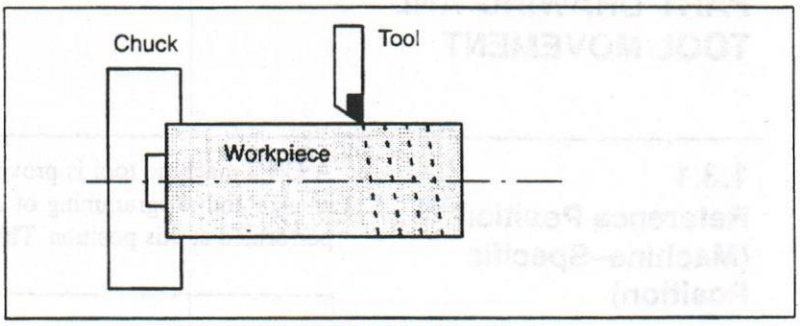


Fig. 1.2 (a) Feed function

Feedrates can be specified by using actual numerics.
For example, the following command can be used to feed the tool 2 mm while the workpiece makes one turn :

F2.0

The function of deciding the feed rate is called the feed function (See II–5).

1.3 PART DRAWING AND TOOL MOVEMENT

1.3.1 Reference Position (Machine–Specific Position)

A CNC machine tool is provided with a fixed position. Normally, tool change and programming of absolute zero point as described later are performed at this position. This position is called the reference position.

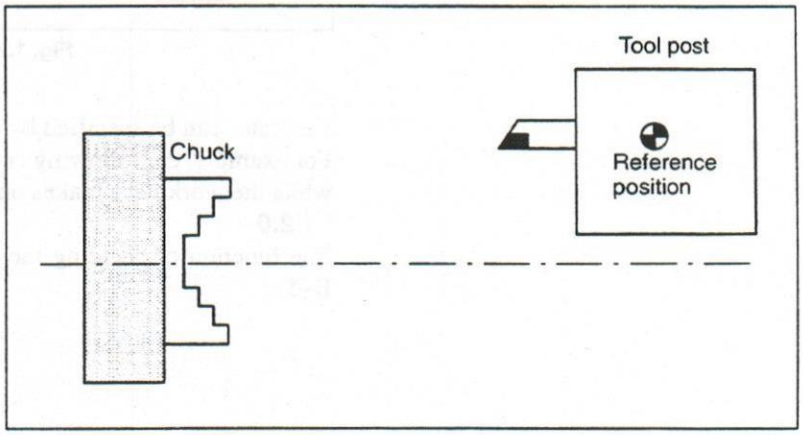


Fig. 1.3.1 (a) Reference position

Explanations

The tool can be moved to the reference position in two ways:

1. Manual reference position return (See III–3.1)
   Reference position return is performed by manual button operation.
2. Automatic reference position return (See II–6)
   In general, manual reference position return is performed first after the power is turned on. In order to move the tool to the reference position for tool change thereafter, the function of automatic reference position return is used.

1.3.2 Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System

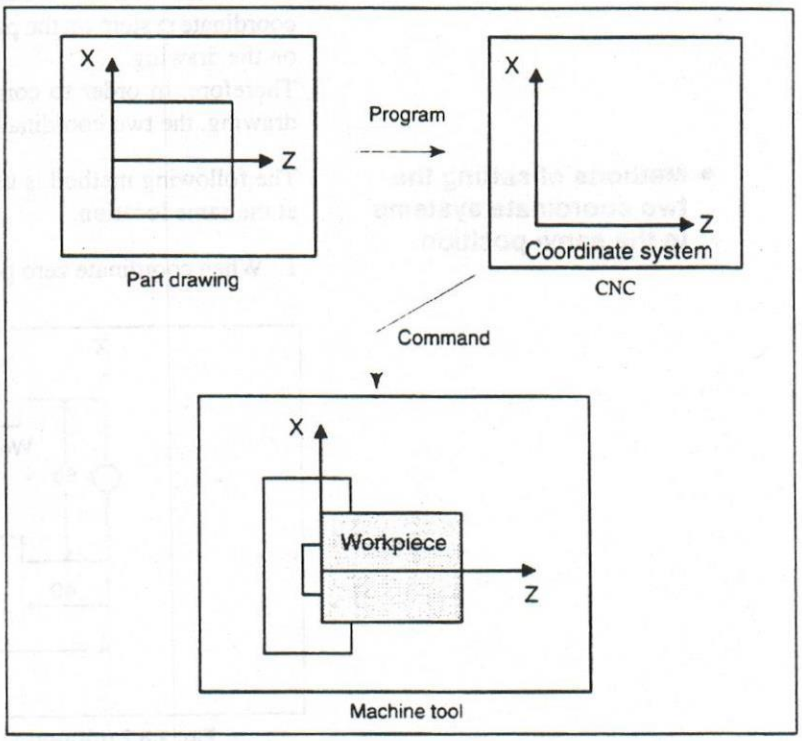


Fig. 1.3.2 (a) Coordinate system

Explanations

- **Coordinate system**

The following two coordinate systems are specified at different locations: (See II–7)

1. Coordinate system on part drawing
   The coordinate system is written on the part drawing. As the program data, the coordinate values on this coordinate system are used.
2. Coordinate system specified by the CNC
   The coordinate system is prepared on the actual machine tool. This can be achieved by programming the distance from the current position of the tool to the zero point of the coordinate system to be set.

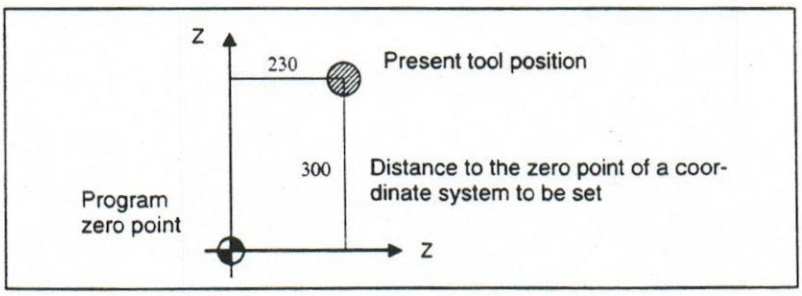


Fig. 1.3.2 (b) Coordinate system specified by the CNC

The tool moves on the coordinate system specified by the CNC in accordance with the command program generated with respect to the coordinate system on the part drawing, and cuts a workpiece into a shape on the drawing.
Therefore, in order to correctly cut the workpiece as specified on the drawing, the two coordinate systems must be set at the same position.

- **Methods of setting the two coordinate systems in the same position**

The following method is usually used to define two coordinate systems at the same location.

1. When coordinate zero point is set at chuck face

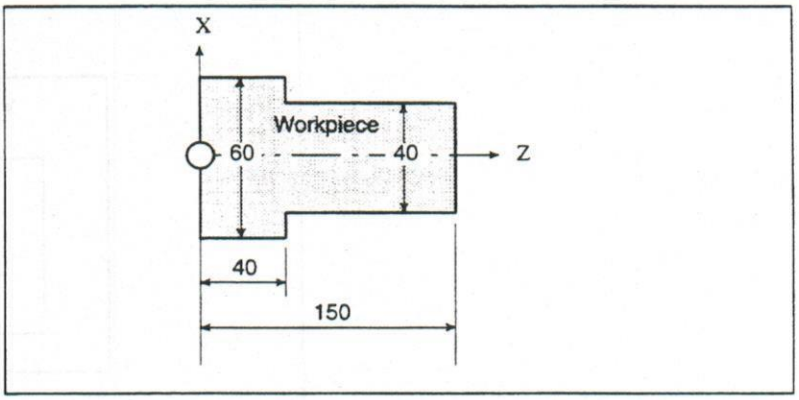


Fig. 1.3.2 (c)Coordinates and dimensions on part drawing

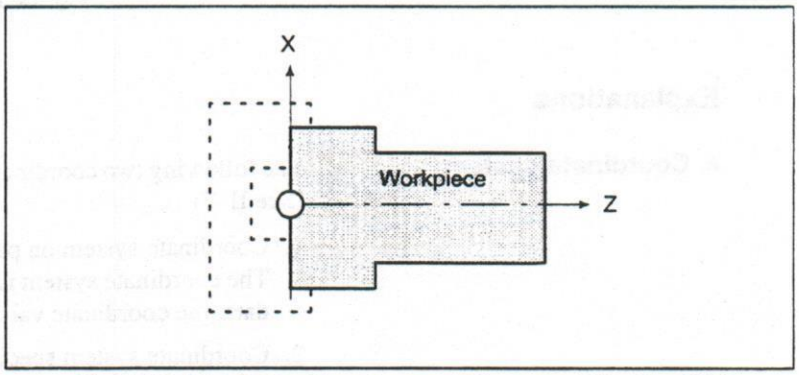


Fig. 1.3.2 (d)Coordinate system on lathe as specified by CNC (made to coincide with the coordinate system on part drawing)

2. When coordinate zero point is set at work end face.

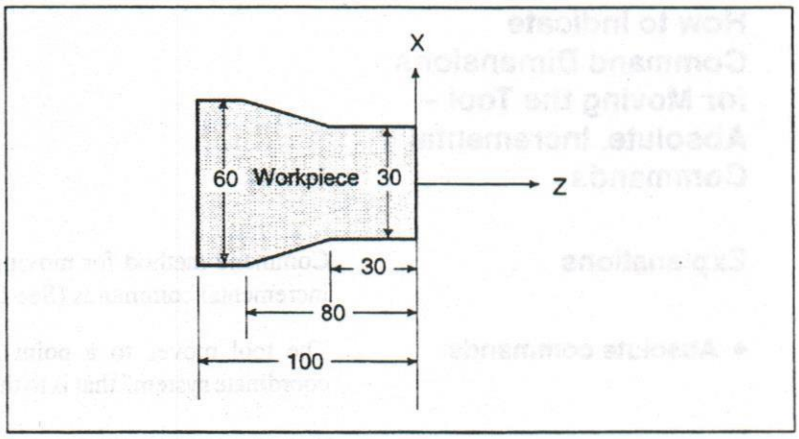


Fig. 1.3.2 (e) Coordinates and dimensions on part drawing

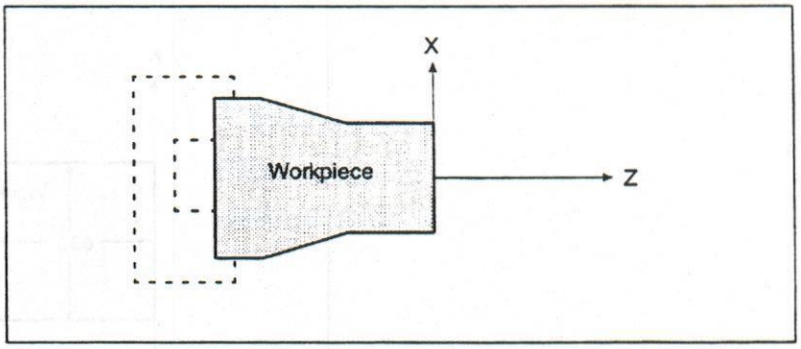


Fig. 1.3.2 (f) Coordinate system on lathe as specified by CNC (made to coincide with the coordinate system on part drawing)

1.3.3 How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands

Explanations

Command method for moving the tool can be indicated by absolute or incremental commands (See II–8.1).

- **Absolute commands**

The tool moves to a point at "the distance from zero point of the coordinate system" that is to the moving position of the coordinate values.

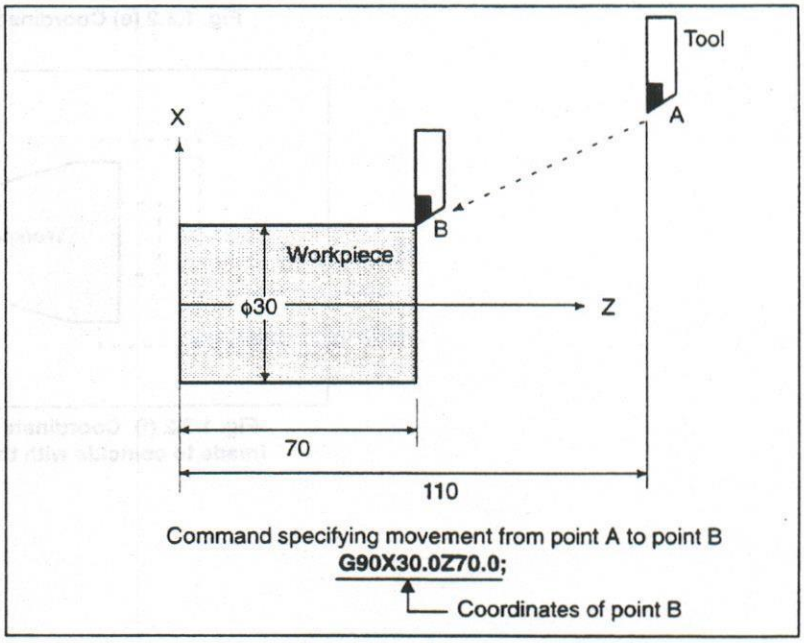


Fig. 1.3.3 (a) Absolute commands

• Incremental commands

Specify the distance from the previous tool position to the next tool position.

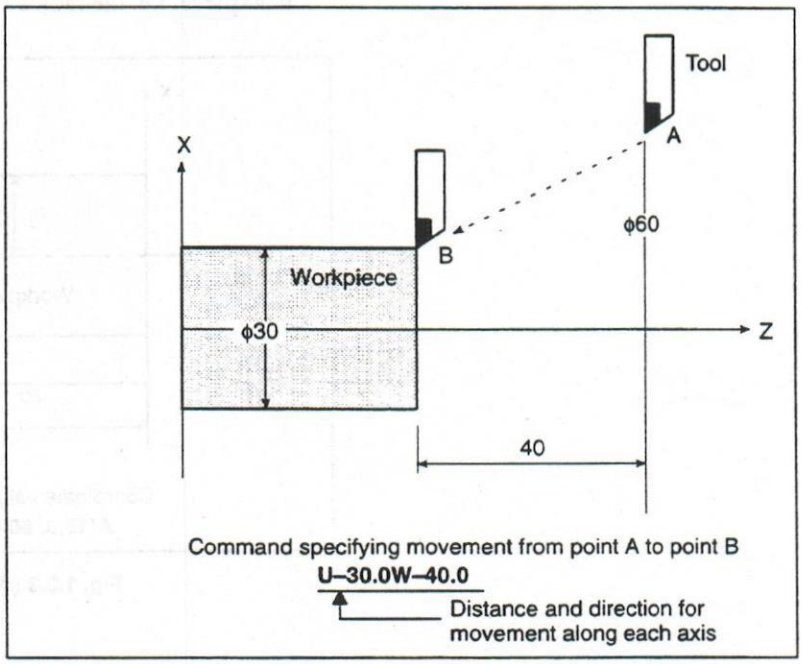


Fig. 1.3.3 (b) Incremental commands

• Diameter programming / radius programming

Dimensions of the X axis can be set in diameter or in radius. Diameter programming or radius programming is employed independently in each machine.

1. Diameter programming
   In diameter programming, specify the diameter value indicated on the drawing as the value of the X axis.

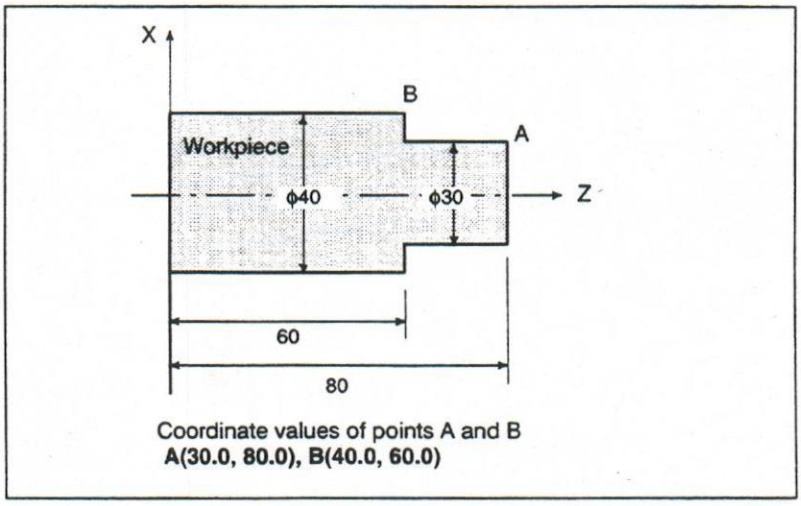


Fig. 1.3.3 (c) Diameter programming

2. Radius programming
   In radius programming, specify the distance from the center of the workpiece, i.e. the radius value as the value of the X axis.

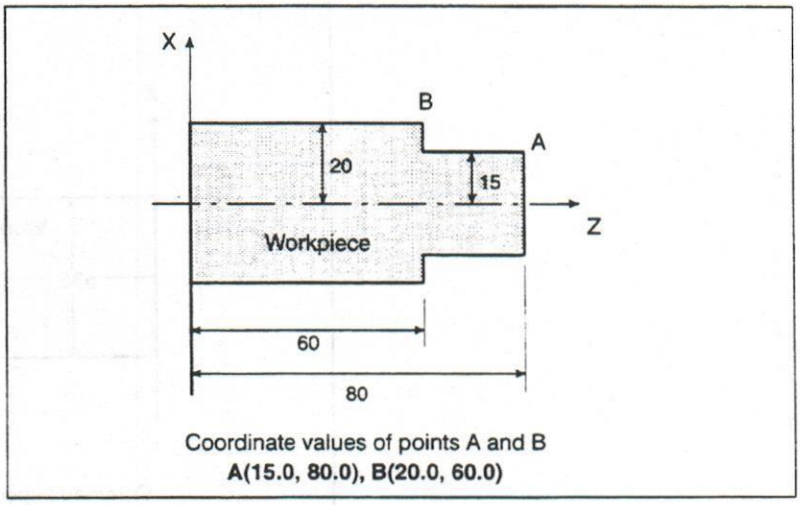


Fig. 1.3.3 (d) Radius programming

1.4 CUTTING SPEED – SPINDLE SPEED FUNCTION

The speed of the tool with respect to the workpiece when the workpiece is cut is called the cutting speed.
As for the CNC, the cutting speed can be specified by the spindle speed in rpm unit.

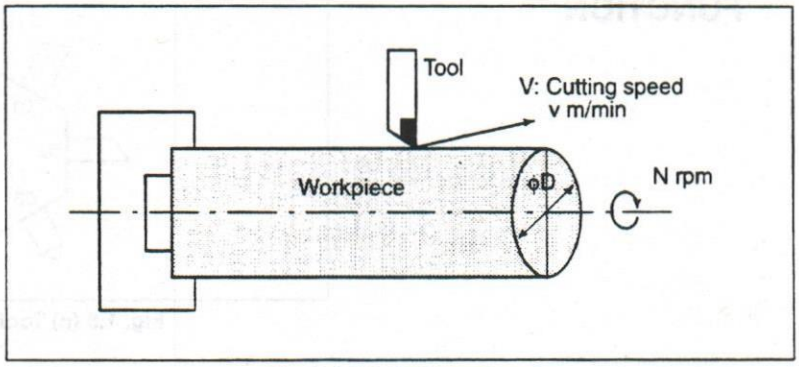


Fig. 1.4 (a) Cutting speed

Examples

<When a workpiece 200 mm in diameter should be machined at a cutting speed of 300 m/min. >

The spindle speed is approximately 478 rpm, which is obtained from $N=1000v/\pi D$. Hence the following command is required:

S478 ;

Commands related to the spindle speed are called the spindle speed function (See II–9).
The cutting speed v (m/min) can also be specified directly by the speed value. Even when the workpiece diameter is changed, the CNC changes the spindle speed so that the cutting speed remains constant.
This function is called the constant surface speed control function (See II–9.3).

1.5 SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION

When drilling, tapping, boring, milling or the like, is performed, it is necessary to select a suitable tool. When a number is assigned to each tool and the number is specified in the program, the corresponding tool is selected.

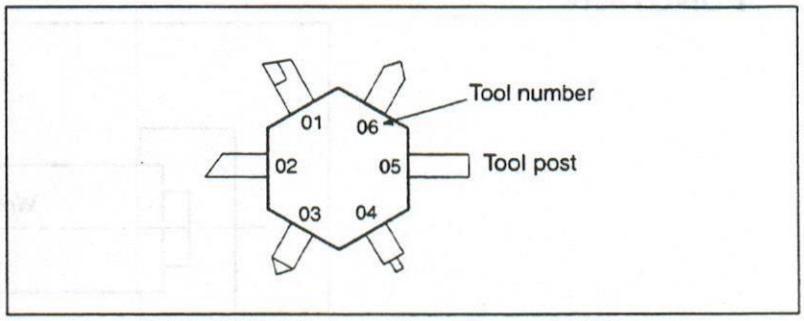


Fig. 1.5 (a) Tool used for various machining

Examples

<When No.01 is assigned to a roughing tool>

When the tool is stored at location 01 of the tool post, the tool can be selected by specifying **T0101**.
This is called the tool function (See II–10).

1.6 COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION

When machining is actually started, it is necessary to rotate the spindle, and feed coolant. For this purpose, on–off operations of spindle motor and coolant valve should be controlled.

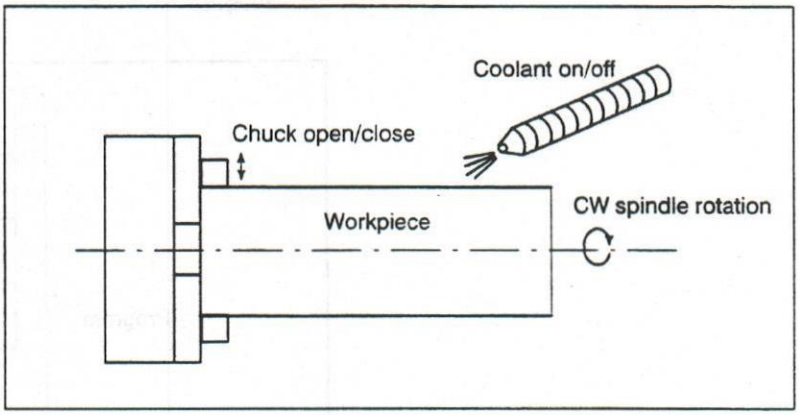


Fig. 1.6 (a) Command for machine operations

The function of specifying the on–off operations of the components of the machine is called the miscellaneous function. In general, the function is specified by an M code. (See II–11)
For example, when M03 is specified, the spindle is rotated clockwise at the specified spindle speed.

1.7 PROGRAM CONFIGURATION

A group of commands given to the CNC for operating the machine is called the program. By specifying the commands, the tool is moved along a straight line or an arc, or the spindle motor is turned on and off.
In the program, specify the commands in the sequence of actual tool movements.

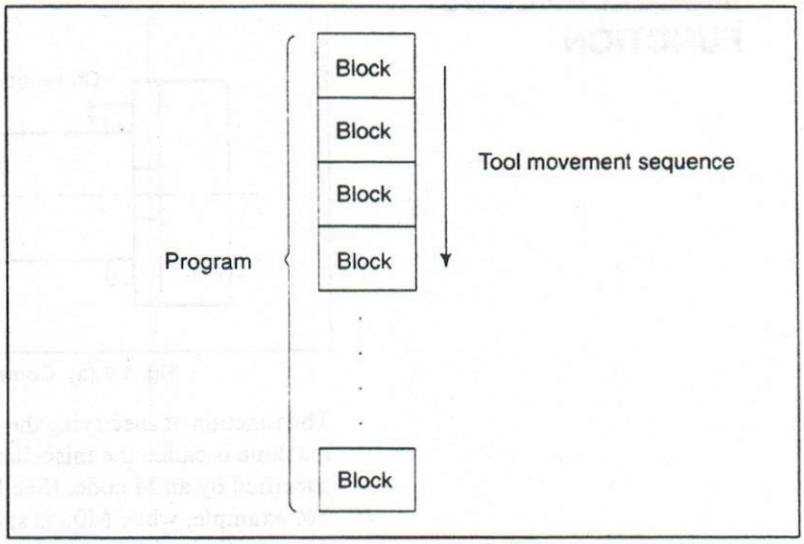


Fig. 1.7 (a) Program configuration

A group of commands at each step of the sequence is called the block. The program consists of a group of blocks for a series of machining. The number for discriminating each block is called the sequence number, and the number for discriminating each program is called the program number (See II–12).

Explanations

The block and the program have the following configurations.

- **Block**

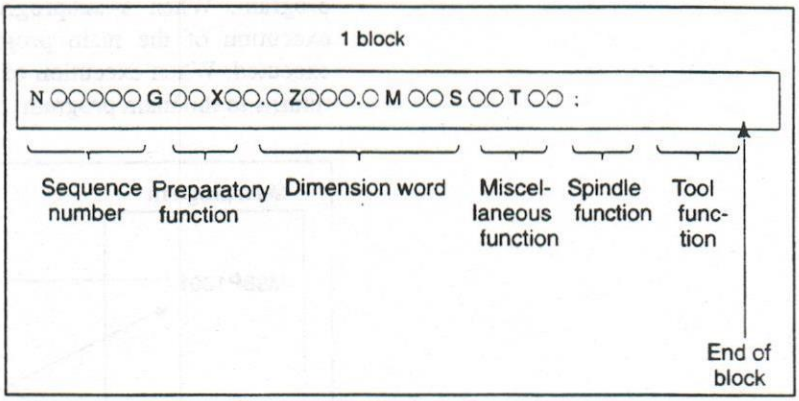


Fig. 1.7 (b) Block configuration

Each block contains a sequence number for identifying the block at the beginning of the block, and an end–of–block code for indicating the end of the block.
This manual indicates the end–of–block code by ; (LF in the ISO code and CR in the EIA code).

- **Program**

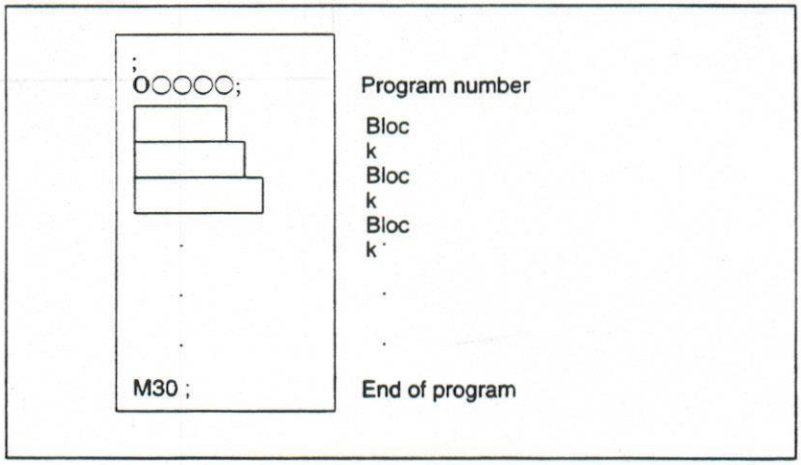
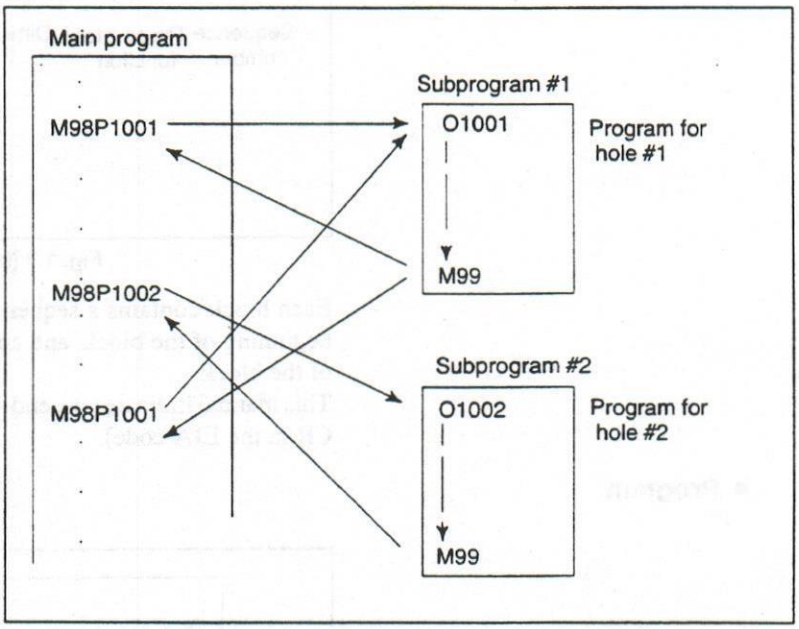


Fig. 1.7 (c) Program configuration

Normally, a program number is specified after the end–of–block (;) code at the beginning of the program, and a program end code (M02 or M30) is specified at the end of the program.

- **Main program and subprogram**

When machining of the same pattern appears at many portions of a program, a program for the pattern is created. This is called the subprogram. On the other hand, the original program is called the main program. When a subprogram execution command appears during execution of the main program, commands of the subprogram are executed. When execution of the subprogram is finished, the sequence returns to the main program.



1.8 TOOL FIGURE AND TOOL MOTION BY PROGRAM

Explanations

- **Machining using the end of cutter – Tool length compensation function (See II–15.1)**

Usually, several tools are used for machining one workpiece. The tools have different tool length. It is very troublesome to change the program in accordance with the tools.
Therefore, the length of each tool used should be measured in advance. By setting the difference between the length of the standard tool and the length of each tool in the CNC (data display and setting : see III–11), machining can be performed without altering the program even when the tool is changed. This function is called tool length compensation. (See II–14)

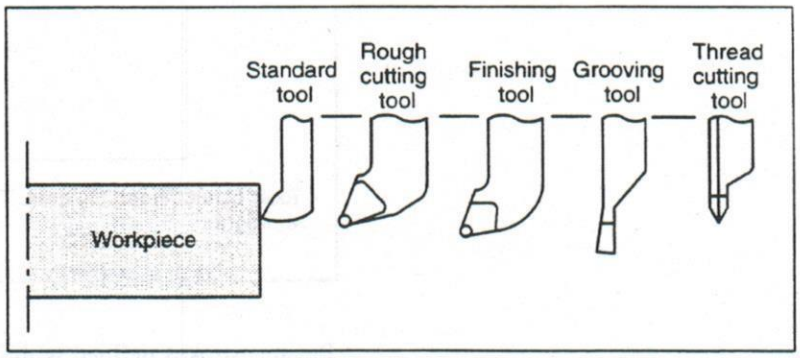
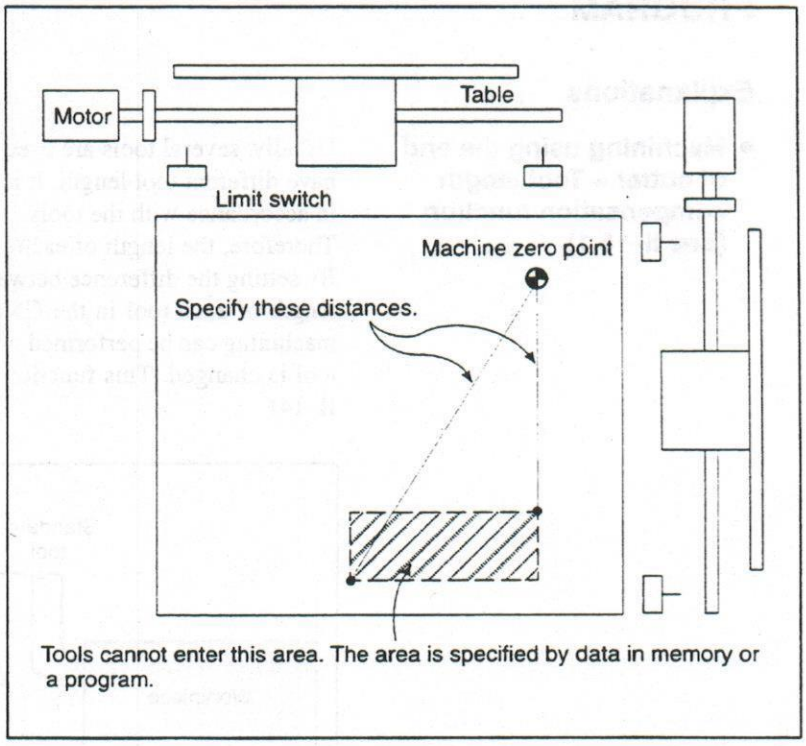


Fig. 1.8 (a) Tool offset

1.9 TOOL MOVEMENT RANGE – STROKE

Limit switches are installed at the ends of each axis on the machine to prevent tools from moving beyond the ends. The range in which tools can move is called the stroke.


Motor
Table
Limit switch
Machine zero point
Specify these distances.
Tools cannot enter this area. The area is specified by data in memory or a program.

Besides strokes defined with limit switches, the operator can define an area which the tool cannot enter using a program or data in memory. This function is called stroke check (See III–6.3).

2 CONTROLLED AXES

2.1 CONTROLLED AXES

Number of controlled axes	2 axes
Simultaneously controlled axis	2 axes

Note
The number of simultaneously controllable axes for manual operation (continuous manual feed, manual reference point return, or manual rapid traverse is 1 or 2 (1 when bit 0 (JAX) of parameter 1002 is set to 0 and 2 when it is set to 1).

2.2 NAMES OF AXES

The first and second axes should be named X and Z, respectively.

2.3 INCREMENT SYSTEM

The increment system consists of the least input increment (for input) and least command increment (for output). The least input increment is the least increment for programming the travel distance. The least command increment is the least increment for moving the tool on the machine. Both increments are represented in mm or inches.

Table 2.3 (a) Increment system IS–B

		Least input increment	Least command increment
Metric system machine	mm input	0.001mm(Diameter)	0.0005mm
		0.001mm(Radius)	0.001mm
	inch input	0.0001inch(Diameter)	0.0005mm
		0.0001inch(Radius)	0.001mm
Inch machine system	mm input	0.001mm(Diameter)	0.00005inch
		0.001mm(Radius)	0.0001inch
	inch input	0.0001inch(Diameter)	0.00005inch
		0.0001inch(Radius)	0.0001inch

The least command increment is either metric or inch depending on the machine tool. Set metric or inch to the parameter INM (No. 100#0).

For selection between metric and inch for the least input increment, G code (G20 or G21) or a setting parameter selects it.

2.4 MAXIMUM STROKES

The maximum stroke controlled by this CNC is shown in the table below:
Maximum stroke=Least command increment±99999999

Table 2.4 (a) Increment system IS–B

Increment system		Maximum strokes
IS–B	Metric machine system	±99999.999 mm
	Inch machine system	±9999.9999 inch

Notes
1. The unit in the table is a diameter value with diameter programming and a radius value in radius programming.
2. A command exceeding the maximum stroke cannot be specified.
3. The actual stroke depends on the machine tool.

3 PREPARATORY FUNCTION (G FUNCTION)

A number following address G determines the meaning of the command for the concerned block.
G codes are divided into the following two types.

Type	Meaning
One–shot G code	The G code is effective only in the block in which it is specified
Modal G code	The G code is effective until another G code of the same group is specified.

(Example)
G01 and G00 are modal G codes.

```
G01X__;
   Z__;   } G01 is effective in this range
   X__;
G00Z__;
```

There are three G code systems : A,B, and C (Table 3). Select a G code system using bits 6 (GSB) and 7 (GSC) of parameter 3401. Generally, this manual describes the use of G code system A, except when the described item can use only G code system B or C. In such cases, the use of G code system B or C is described.

Explanations

1.If the CNC enters the clear state (see bit 6 (CLR) of parameter 3402) when the power is turned on or the CNC is reset, the modal G codes change as follows.

(1) G codes marked with ◤ in Table 3 are enabled.

(2) When the system is cleared due to power–on or reset,whichever specified, either G20 or G21, remains effective.

(3) Setting bit 0 (G01) of parameter 3402 determines which code, either G00 or G01, is effective.

(4) In G code system B or C, setting bit 3 (G91) of parameter 3402 determines which code, either G90 or G91, is effective.

2.G codes of group 00 except G10 and G11 are single–shot G codes.

3.Alarm 010 is displayed when a G code not listed in the G code list is specified or a G code without a corresponding option is specified.

4.G codes of different groups can be specified in the same block. If G codes of the same group are specified in the same block, the G code specified last is valid.

5.G codes are displayed for each group number.

6.In G code system A, absolute commands are discriminated from incremental commands by address words, instead of G codes (G90/G91). (See Section 8.1 "Absolute and incremental programming".)

Table 3 G code list

G code			Group	Function
A	B	C		
G00	G00	G00	01	Positioning (Rapid traverse)
G01	G01	G01		Linear interpolation (Cutting feed)
G02	G02	G02		Circular interpolation CW
G03	G03	G03		Circular interpolation CCW
G04	G04	G04	00	Dwell
G10	G10	G10		Data setting
G11	G11	G11		Data setting mode (programmable parameter input)
G20	G20	G70	06	Input in inch
G21	G21	G71		Input in mm
G27	G27	G27	00	Reference position return check
G28	G28	G28		Return to reference position
G30	G30	G30		2nd reference position return
G31	G31	G31		Skip function
G32	G33	G33	01	Thread cutting
G40	G40	G40	07	Tool nose radius compensation cancel
G41	G41	G41		Tool nose radius compensation left
G42	G42	G42		Tool nose radius compensation right
G50	G92	G92	00	Coordinate system setting or max. spindle speed setting
G65	G65	G65		Macro calling
G66	G66	G66	12	Macro modal call
G67	G67	G67		Macro modal call cancel
G90	G77	G20	01	Outer diameter/internal diameter cutting cycle
G92	G78	G21		Thread cutting cycle
G94	G79	G24		Endface turning cycle
G96	G96	G96	02	Constant surface speed control
G97	G97	G97		Constant surface speed control cancel
G98	G94	G94	05	Per minute feed
G99	G95	G95		Per revolution feed
–	G90	G90	03	Absolute programming
–	G91	G91		Incremental programming

4 INTERPOLATION FUNCTIONS

4.1 POSITIONING (G00)

The G00 command moves a tool to the position in the workpiece system specified with an absolute or an incremental command at a rapid traverse rate.
In the absolute command, coordinate value of the end point is programmed.
In the incremental command the distance the tool moves is programmed.

Format

G00 IP_;

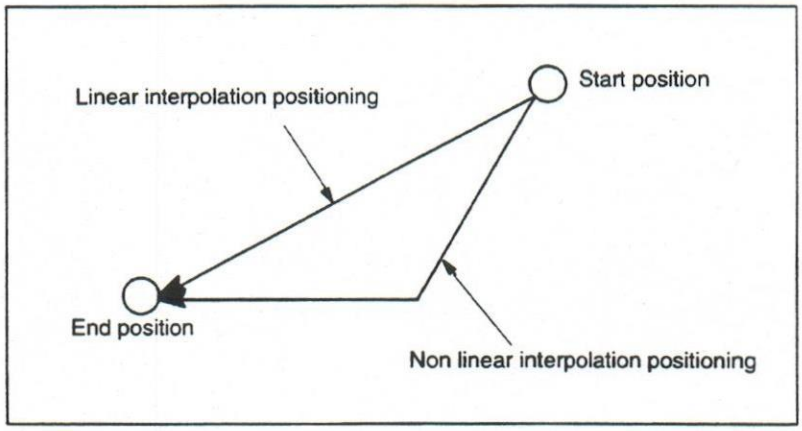
IP_: For an absolute command, the coordinates of an end position, and for an incremental command, the distance the tool moves.

Explanations

Either of the following tool paths can be selected according to bit 1 of parameter LRP (No. 1401).

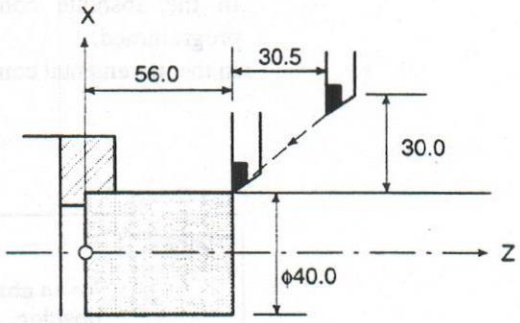
- **Nonlinear interpolation positioning**
  The tool is positioned with the rapid traverse rate for each axis separately. The tool path is normally straight.

- **Linear interpolation positioning**
  The tool path is the same as in linear interpolation (G01). The tool is positioned within the shortest possible time at a speed that is not more than the rapid traverse rate for each axis.



The rapid traverse rate in the G00 command is set to the parameter No.1420 for each axis independently by the machine tool builder. In the positioning mode actuated by G00, the tool is accelerated to a predetermined speed at the start of a block and is decelerated at the end of a block. Execution proceeds to the next block after confirming the in–position.
"In–position" means that the feed motor is within the specified range. This range is determined by the machine tool builder by setting to parameter No.1826. The user can disable in-position check for each block by setting bit 5 (NCI) of parameter No.1601 accordingly.

Examples



< Radius programming >
G00X40.0Z56.0 ; (Absolute command)
or
G00U–60.0W–30.5;(Incremental command)

Restrictions

The rapid traverse rate cannot be specified in the address F.
Even if linear interpolation positioning is specified, nonlinear interpolation positioning is used in the following cases. Therefore, be careful to ensure that the tool does not foul the workpiece.

- G28 specifying positioning between the reference and intermediate positions.

4.2 LINEAR INTERPOLATION (G01)

Tools can move along a line.

Format

G01 IP_ F_;

IP_: For an absolute command, the coordinates of an end point , and for an incremental command, the distance the tool moves.

F_: Speed of tool feed (Feedrate)

Explanations

A tools move along a line to the specified position at the feedrate specified in F.

The feedrate specified in F is effective until a new value is specified. It need not be specified for each block.

The feedrate commanded by the F code is measured along the tool path. If the F code is not commanded, the feedrate is regarded as zero.

For feed–per–minute mode under 2–axis simultaneous control, the feedrate for a movement along each axis as follows :

G01 $\alpha\_\beta\_$ F_f ;

Feed rate of α axis direction : $Fa = \frac{\alpha}{L} \times f$

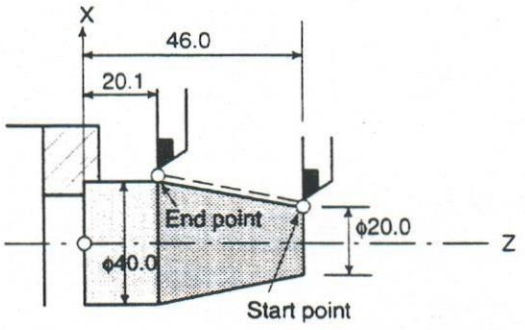
Feed rate of B axis direction : $F_\beta = \frac{\beta}{L} \times f$

$L = \sqrt{\alpha^2 + \beta^2 + \gamma^2 + \zeta^2}$

Examples

- **Linear interpolation**

< Diameter programming >
G01X40.0Z20.1F20 ; (Absolute command)
or
G01U20.0W–25.9F20 ; (Incremental command)



4.3 CIRCULAR INTERPOLATION (G02,G03)

The command below will move a tool along a circular arc on the ZX plane.

Format

Arc in the ZX plane

$$\left\{\begin{matrix}\text{G02}\\\text{G03}\end{matrix}\right\} \text{X(U)\_Z(W)\_} \left\{\begin{matrix}\text{I\_K\_}\\\text{R\_}\end{matrix}\right\} \text{F\_;}$$

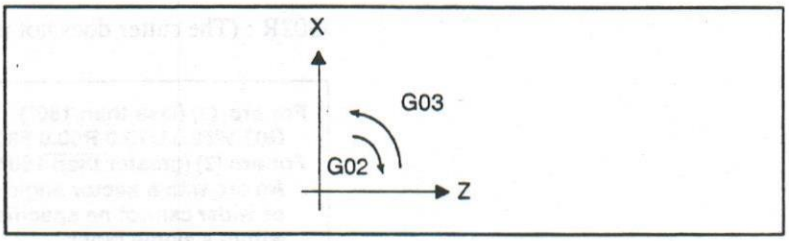
Table.4.3 Description of the Command Format

Command	Description
G02	Circular Interpolation Clockwise direction (CW)
G03	Circular Interpolation Counterclockwise direction (CCW)
X_(U_)	Command values of X axis
Z_(W_)	Command values of Z axis
I_	X axis distance from the start point to the center of an arc (with sign, radius programming)
K_	Z axis distance from the start point to the center of an arc (with sign, radius programming)
R_	Arc radius (with no sign, radius programming)
F_	Feedrate along the arc

Explanations

- **Direction of the circular interpolation**

"Clockwise"(G02) and "counterclockwise"(G03) on the X_pY_p plane (Z_pX_p plane or Y_pZ_p plane) are defined when the X_pY_p plane is viewed in the positive–to–negative direction of the Z_p axis (Y_p axis or X_p axis, respectively) in the Cartesian coordinate system. See the figure below.

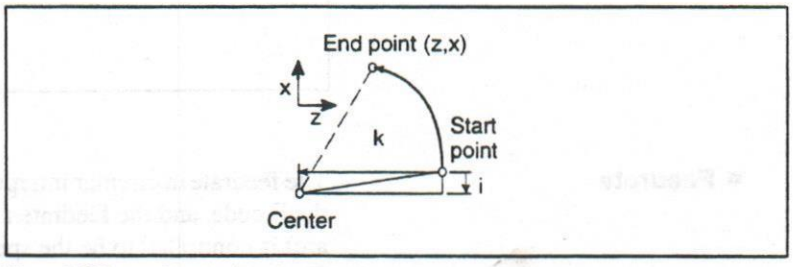


- **Distance moved on an arc**

The end point of an arc is specified by address X or Z, and is expressed as an absolute or incremental value according to G90 or G91. For the incremental value, the distance of the end point which is viewed from the start point of the arc is specified.

- **Distance from the start point to the center of arc**

The arc center is specified by addresses I and K for the X and Z axes, respectively. The numerical value following I or K, however, is a vector component in which the arc center is seen from the start point, and is always specified as an incremental value irrespective of G90 and G91, as shown below.
I and K must be signed according to the direction.



I0 and K0 can be omitted.
If the difference between the radius at the start point and that at the end point exceeds the value in a parameter (No.3410), an alarm (No.020) occurs.

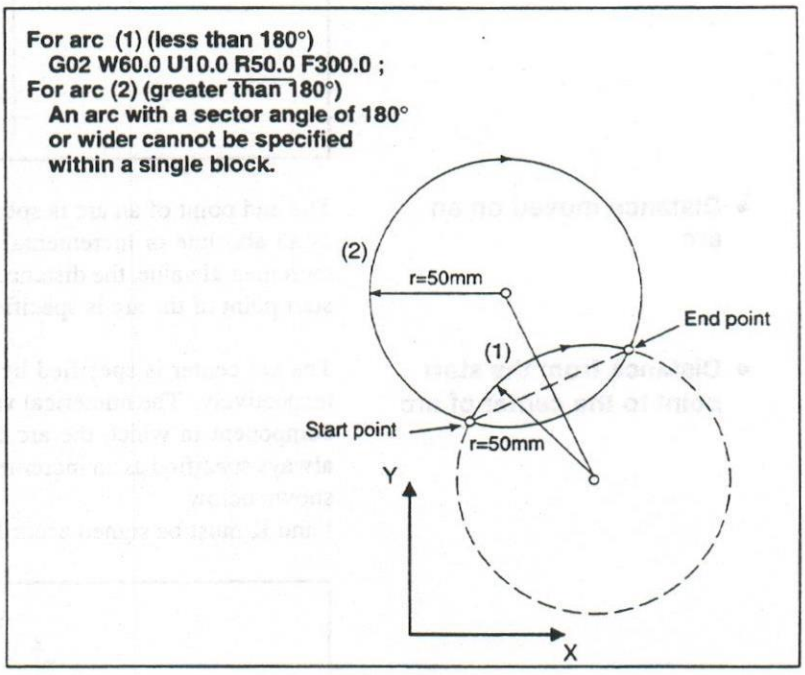
- **Full–circle programming**

When X and Z are omitted (the end point is the same as the start point) and the center is specified with I and K, a 360° arc (circle) is specified.
<Example> G02 I_; (circle)

• Arc radius

The distance between an arc and the center of a circle that contains the arc can be specified using the radius, R, of the circle instead of I and K.
In this case, one arc is less than 180°, and the other is more than 180° are considered. An arc with a sector angle of 180°or wider cannot be specified. If X and Z are all omitted, if the end point is located at the same position as the start point and when R is used, an arc of 0°is programmed

G02R ; (The cutter does not move.)



• Feedrate

The feedrate in circular interpolation is equal to the feed rate specified by the F code, and the feedrate along the arc (the tangential feedrate of the arc) is controlled to be the specified feedrate.
The error between the specified feedrate and the actual tool feedrate is ±2% or less. However, this feed rate is measured along the arc after the tool nose radius compensation is applied

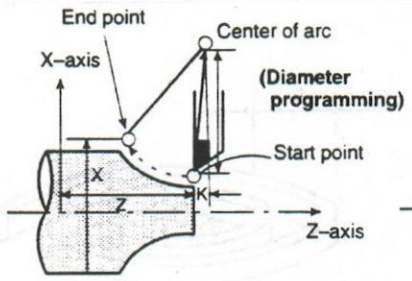
Restrictions

If I, K, and R addresses are specified simultaneously, the arc specified by address R takes precedence and the other are ignored.
If the difference in the radius between the start and end points of the arc exceeds the value specified in parameter No. 3410, alarm No. 020 is generated.
If the end point is not on the arc, the tool moves in a straight line along one of the axes after reaching the end point.
When an arc having a center angle close to 180ϕ is specified using its radius R, the system may fail to calculate the center of the arc correctly. Therefore, specify the arc with I, J, and K.

Examples

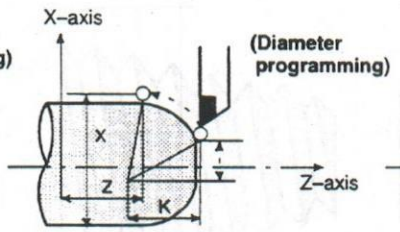
- **Command of circular interpolation X, Z**

G02X_Z_I_K_F_;



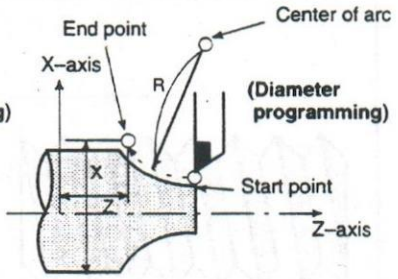
(Absolute programming)

G03X_Z_I_K_F_;

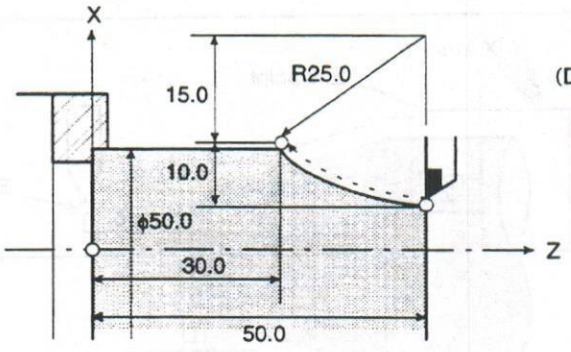


(Absolute programming)

G02X_Z_R_F_;



(Absolute programming)



(Diameter programming)
G02X50.0Z30.0I25.0F0.3; or
G02U20.0W–020.0I25.0F0.3; or
G02X50.0Z30.0R25.0F0.3 or
G02U20.0W–20.0R25.F0.3;

4.4 CONSTANT LEAD THREADING (G32)

Tapered screws and scroll threads in addition to equal lead straight threads can be cut by using a G32 command.
The spindle speed is read from the position coder on the spindle in real time and converted to a cutting feedrate for feed–per minute mode, which is used to move the tool.

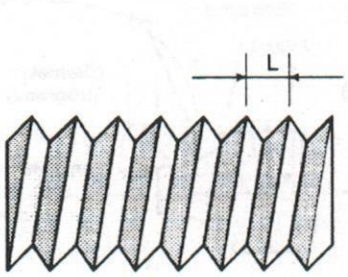


Fig. 4.4 (a) Straight Thread

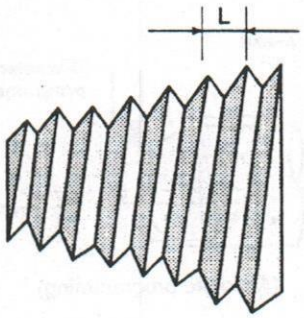


Fig. 4.4 (b) Tapered Screw

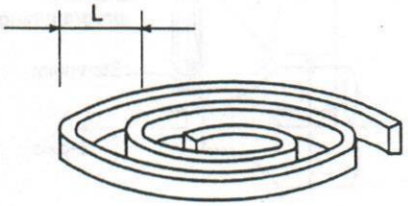


Fig. 4.4 (c) Scroll Thread

Format

G32 IP_ F_;

IP_ : End point
F_ : Lead of the long axis (always radius programming)

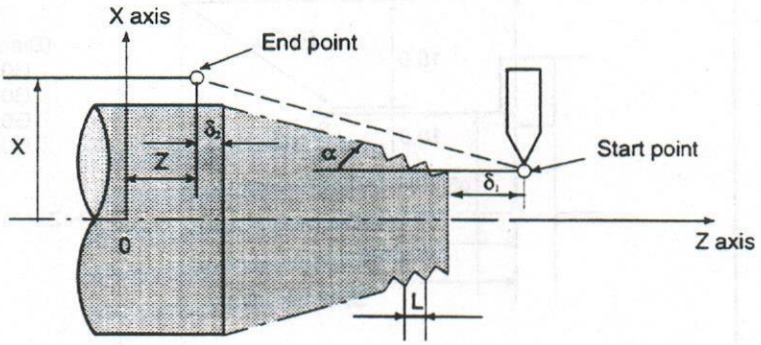


Fig. 4.4 (d) Example of Thread Cutting

Explanations

In general, thread cutting is repeated along the same tool path in rough cutting through finish cutting for a screw. Since thread cutting starts when the position coder mounted on the spindle outputs a 1–turn signal, threading is started at a fixed point and the tool path on the workpiece is unchanged for repeated thread cutting. Note that the spindle speed must remain constant from rough cutting through finish cutting. If not, incorrect thread lead will occur.

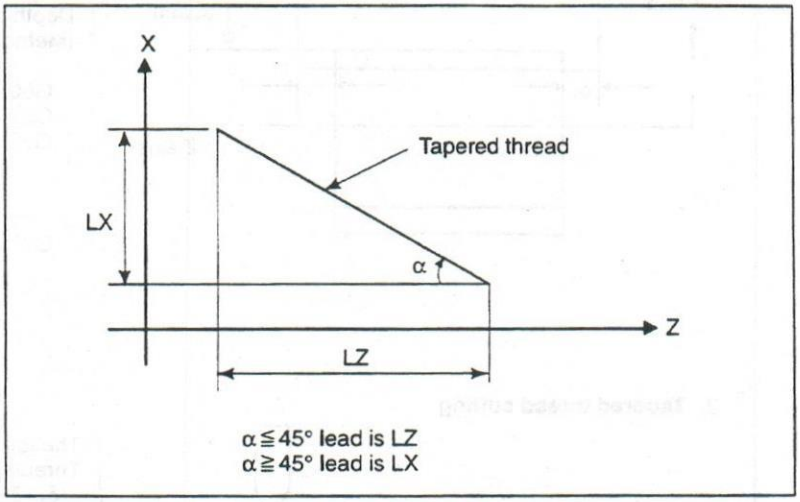


Fig. 4.4 (e) LZ and LX of a Tapered Thread

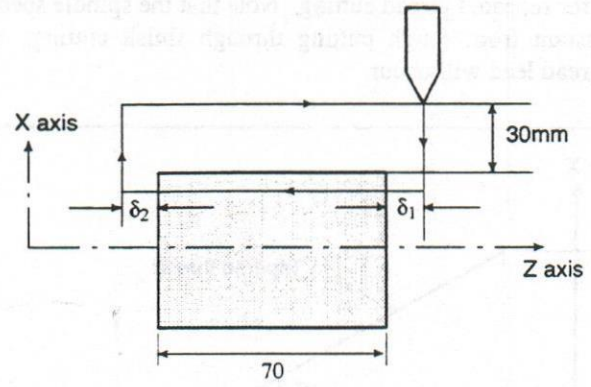
In general, the lag of the servo system, etc. will produce somewhat incorrect leads at the starting and ending points of a thread cut. To compensate for this, a threading length somewhat longer than required should be specified.
Table 4.4 (a) lists the ranges for specifying the thread lead.

Table. 4.4 (a) Ranges of lead sizes that can be specified

	Least command increment
mm input	0.0001 A500.0000mm
Inch input	0.000001 inchA9.999999inch

Explanations

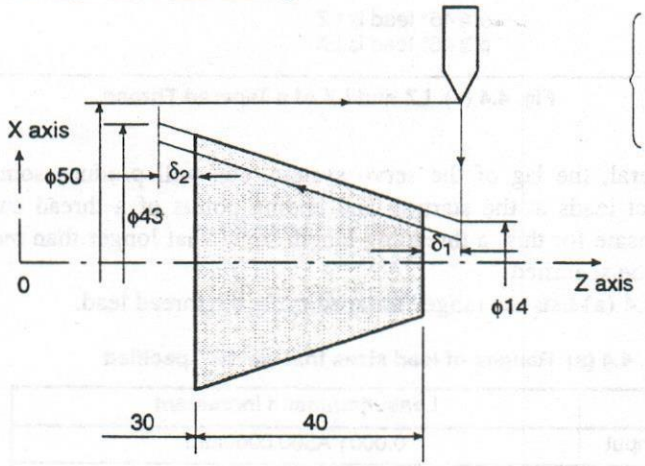
1. Straight thread cutting



The following values are used in programming :
Thread lead :4mm
δ_1=3mm
δ_2=1.5mm
Depth of cut :1mm (cut twice)
(Metric input, Diameter programming)

```
G00  U–62.0 ;
G32  W–74.5 F4.0 ;
G00  U62.0 ;
     W74.5 ;
     U–64.0 ;
     (For the second cut, cut 1mm more)
G32  W–74.5 ;
     G00  U64.0 ;
     W74.5 ;
```

2. Tapered thread cutting



The following values are used in programming :
Thread lead : 3.5mm in the direction of the Z axis
δ_1=2mm
δ_2=1mm
Cutting depth in the X axis direction is 1mm
(Cut twice)
(Metric input, Diameter programming)

```
G00  X 12.0 Z72.0 ;
G32  X 41.0 Z29.0 F3.5 ;
G00  X 50.0 ;
     Z 72.0 ;
     X 10.0 ;
     (Cut 1mm more for the second cut)
G32  X 39.0 Z29.0 ;
G00  X 50.0 ;
     Z 72.0 ;
```

Notes

1. Feedrate override is effective (fixed at 100%) during thread cutting.
2. It is very dangerous to stop feeding the thread cutter. This will suddenly increase the cutting depth. Thus, the feed hold function is ineffective while thread cutting.If the feed hold button is pressed during thread cutting, the tool stop after the first block not specifying thread cutting is executed as if the SINGLE BLOCK button were pushed. However, the feed lamp (SPL lamp) lights when FEED HOLD button on the machine control panel is pushed. Then, when the tool stops, the lamp is tumed off (Single Block stop status).
3. When the FEED HOLD button is pressed again during the execution of the first non–thread cutting block after a thread cutting mode (or the button is held down), the tool stops immediately.
4. When thread cutting is executed in the single block status, the tool stops after the execution of the first non–thread cutting block that follows the thread cutting mode.
5. When the mode was changed from automatic operation to manuale operation during thread cutting, the tool stops at the first non–thread cutting block not specifying thread cutting as when the feed hold button is pushed as mentioned in Note 3.

   However, when the mode is changed from one automatic operation mode to another, the tool stops after execution of the block not specifying thread cutting as for the single block mode in Note. 4.
6. When the previous block was a thread cutting block, cutting will start immediately without waiting for detection of the 1–turn signal even if the present block is a thread cutting block. The thread can not be cut correctly between the blocks.

   G32Z_ F_;

   Z_; (A 1–turn signal is not detected before this block.)

   G32 ; (Regarded as threading block.)

   Z_ F_; (One turn signal is also not detected.)
7. When constant surface speed control is put in effect during scroll or tapered thread cutting, the spindle speed changes, possibly disabling the correct screw lead from being cut. Therefore, do not use the constant surface speed control during thread cutting. Instead, use G97.
8. A movement block preceding the thread cutting block must not specify chamfering or corner R.
9. A thread cutting block must not specifyung chamfering or corner R.
10. The spindle speed override function is disabled during thread cutting. The spindle speed is fixed at 100%.

4.5 MULTIPLE-THREAD CUTTING

Using the Q address to specify an angle between the one-spindle-rotation signal and the start of threading shifts the threading start angle, making it possible to produce multiple-thread screws with ease.

Format

(constant-lead threading)

G32 IP_ F_ Q_ ;
G32 IP_ Q_ ;

IP_ : End point
F_ : Lead in longitudinal direction
Q_ : Threading start angle

Explanations

- **Available thread cutting commands**

G32: Constant-lead thread cutting
G92: Thread cutting cycle

Limitations

- **Start angle**

The start angle is not a continuous-state (modal) value. It must be specified each time it is used. If a value is not specified, 0 is assumed.

- **Start angle increment**

The start angle (Q) increment is 0.001 degrees. Note that no decimal point can be specified.
Example:
For a shift angle of 180 degrees, specify Q180000.
Q180.000 cannot be specified, because it contains a decimal point.

- **Specifiable start angle range**

A start angle (Q) of between 0 and 360000 (in 0.001-degree units) can be specified. If a value greater than 360000 (360 degrees) is specified, it is rounded down to 360000 (360 degrees).

Examples

Program for producing double-threaded screws (with start angles of 0 and 180 degrees)

```
G00 X40.0 ;
G32 W-38.0 F4.0 Q0 ;
G00 X72.0 ;
    W38.0 ;
    X40.0 ;
G32 W-38.0 F4.0
Q180000 ;
G00 X72.0 ;
    W38.0 ;
```

4.6 SKIP FUNCTION (G31)

Linear interpolation can be commanded by specifying axial move following the G31 command, like G01. If an external skip signal is input during the execution of this command, execution of the command is interrupted and the next block is executed.
The skip function is used when the end of machining is not programmed but specified with a signal from the machine, for example, in grinding. It is used also for measuring the dimensions of a workpiece.
For details of how to use this function, refer to the manual supplied by the machine tool builder.

Format

G31 IP_ ;

G31: One-shot G code (If is effective only in the block in which it is specified)

Explanations

The coordinate values when the skip signal is turned on can be used in a custom macro because they are stored in the custom macro system variable #5061 to #5062, as follows:

#5061 X axis coordinate value
#5062 Z axis coordinate value

Notes

1. If G31 command is issued while tool nose radius compensation is applied, an P/S alarm of No.035 is displayed. Cancel the cutter compensation with the G40 command before the G31 command is specified.
2. If the feedrate per minute is specified, feedrate override, dry run, and automatic acceleration/deceleration are disabled during movement of the tool triggered by the skip function. This is intended to improve the tool positioning accuracy when a skip signal is input. These functions are enabled if the feedrate per rotation is specified. However, they can be enabled by bit 7 (SKF) of parameter No. 6200.

Examples

- **The next block to G31 is an incremental command**

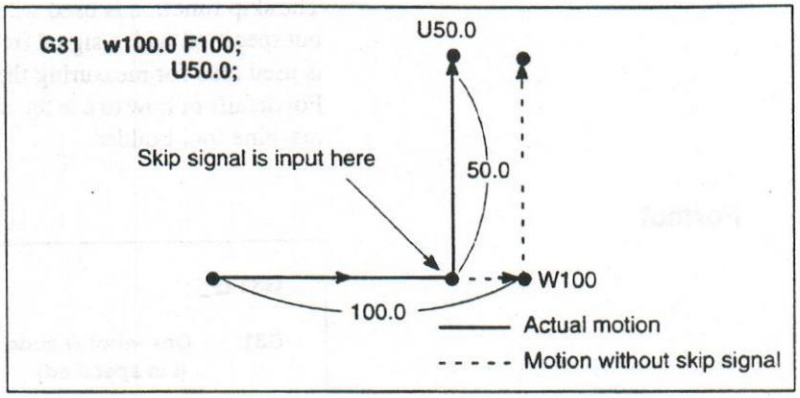


Fig.4.6 (a) The next block is an incremental command

- **The next block to G31 is an absolute command for 1 axis**

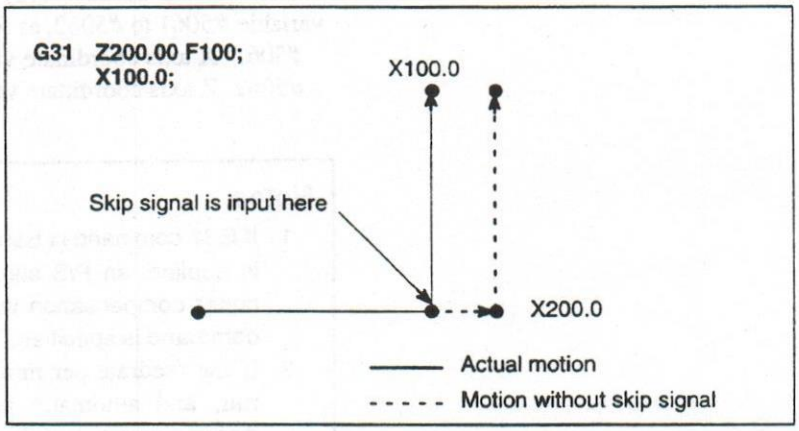


Fig.4.6 (b) The next block is an absolute command for 1 axis

- **The next block to G31 is an absolute command for 2 axes**

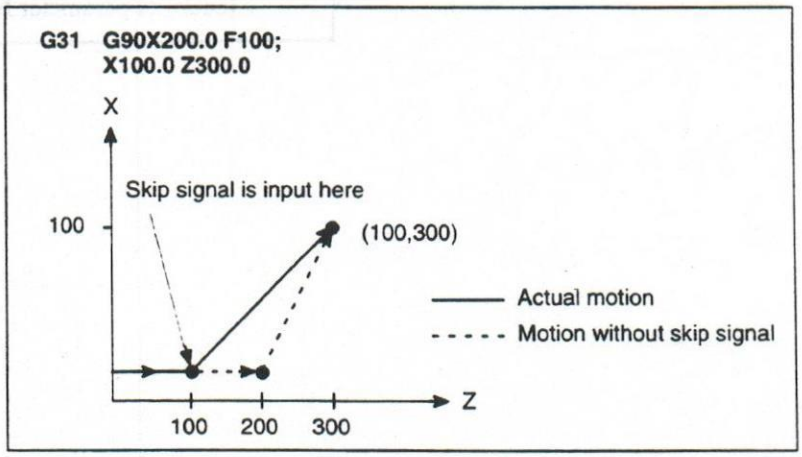


Fig 4.6 (c) The next block is an absolute command for 2 axes

5 FEED FUNCTIONS

5.1 GENERAL

The feed functions control the feedrate of the tool. The following two feed functions are available:

- **Feed functions**

1. Rapid traverse
   When the positioning command (G00) is specified, the tool moves at a rapid traverse feedrate set in the CNC (parameter No. 1420).
2. Cutting feed
   The tool moves at a programmed cutting feedrate.

- **Override**

Override can be applied to a rapid traverse rate or cutting feedrate using the switch on the machine operator's panel.

- **Automatic acceleration/deceleration**

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement.

5.2 RAPID TRAVERSE

Format

```
G00 IP_ ;
    G00 : G code (group 01) for positioning (rapid traverse)
    IP_ ; Dimension word for the end point
```

Explanations

The positioning command (G00) positions the tool by rapid traverse. In rapid traverse, the next block is executed after the specified feedrate becomes 0 and the servo motor reaches a certain range set by the machine tool builder (in–position check).

However, bit 5 (NCI) of parameter No. 1601 can specify that a position check be not performed in every positioning block.

A rapid traverse rate is set for each axis by parameter No. 1420, so no rapid traverse feedrate need be programmed.

The following overrides can be applied to a rapid traverse rate with the switch on the machine operator's panel:F0, 25, 50, 100%

F0: Allows a fixed feedrate to be set for each axis by parameter No. 1421.

For detailed information, refer to the appropriate manual of the machine tool builder.

5.3 CUTTING FEED

Feedrate of linear interpolation (G01), circular interpolation (G02, G03), etc. are commanded with numbers after the F code.
In cutting feed, the next block is executed so that the feedrate change from the previous block is minimized.
Two modes of specification are available:

1. Feed per minute (G98)
   After F, specify the amount of feed of the tool per minute.
2. Feed per revolution (G99)
   After F, specify the amount of feed of the tool per spindle revolution.
3. Fl–digit feed
   Specify a desired one–digit number after F. Then, the feedrate set with the CNC for that number is set.

Format

Feed per minute
G98 ; G code (group 05) for feed per minute
F_ ; Feedrate command (mm/min or inch/min)

Feed per revolution
G99 ; G code (group 05) for feed per revolution
F_ ; Feedrate command (mm/rev or inch/rev)

Explanations

- **Tangential speed constant control**

Cutting feed is controlled so that the tangential feedrate is always set at a specified feedrate.

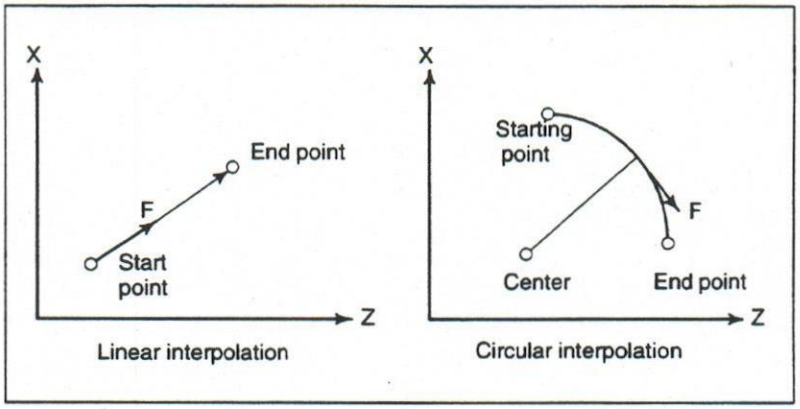


Fig. 5.3 (a) Tangential feedrate (F)

- **Feed per minute (G98)**

After specifying G98 (in the feed per minute mode), the amount of feed of the tool per minute is to be directly specified by setting a number after F. G98 is a modal code. Once a G98 is specified, it is valid until G99 (feed per revolution) is specified. At power–on, the feed per revolution mode is set.
An override from 0% to 254% (in 1% steps) can be applied to feed per minute with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

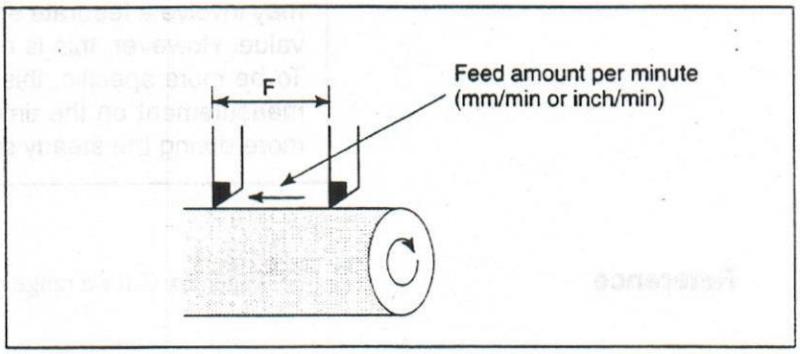


Fig. 5.3 (b) Feed per minute

Note
No override can be used for some commands such as for threading.

- **Feed per revolution (G99)**

After specifying G99 (in the feed per revolution mode), the amount of feed of the tool per spindle revolution is to be directly specified by setting a number after F. G99 is a modal code. Once a G99 is specified, it is valid until G98 (feed per minute) is specified.
An override from 0% to 254% (in 1% steps) can be applied to feed per revolution with the switch on the machine operator's panel. For detailed information, see the appropriate manual of the machine tool builder.

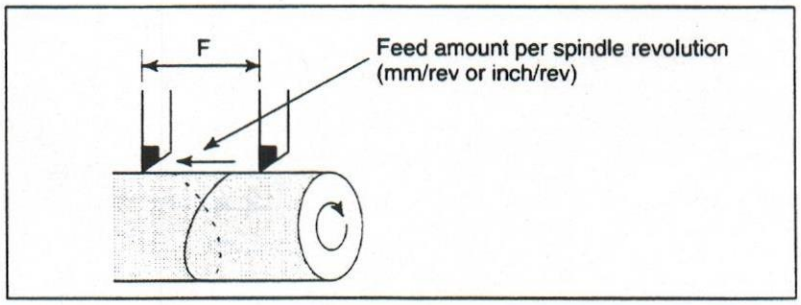


Fig. 5.3 (c) Feed per revolution

Note
When the speed of the spindle is low, feedrate fluctuation may occur. The slower the spindle rotates, the more frequently feedrate fluctuation occurs.

- **Cutting feedrate clamp**

A common upper limit can be set on the cutting feedrate along each axis with parameter No. 1422. If an actual cutting feedrate (with an override applied) exceeds a specified upper limit, it is clamped to the upper limit.

Note
An upper limit is set in mm/min or inch/min. CNC calculation may involve a feedrate error of ±2% with respect to a specified value. However, this is not true for acceleration/deceleration. To be more specific, this error is calculated with respect to a measurement on the time the tool takes to move 500 mm or more during the steady state:

Reference

See Appendix C for a range of feedrates that can be specified.

5.4 DWELL (G04)

Format

Dwell	**G04 X_ ;** or **G04 U_ ;** or **G04 P_ ;** **X_** : Time or number of spindle revokutions (decimal point permitted) **U_** : Time or number of spindle revokutions (decimal point permitted) **P_** : Time or number of spindle revokutions (decimal point not permitted)

Explanations

By specifying a dwell, the execution of the next block is delayed by the specified time.
Machining can be delayed to go to the next block until the spindle rotates by the specified number of revolutions in the feed per revolution mode (G99) by parameter DWL (No.3405#1).

Table 5.4 (a) Command value range of the dwell time (Command by X or U)

Increment system	Command value range	Dwell time unit
IS–B	0.001 to 99999.999	s or rev

Table 5.4 (b) Command value range of the dwell time (Command by P)

Increment system	Command value range	Dwell time unit
IS–B	1 to 99999999	0.001 s or rev

5.5 AUTOMATIC ACCELERATION/ DECELERATION

5.5.1 Automatic Acceleration/ Deceleration

General

· Automatic acceleration/deceleration

To prevent a mechanical shock, acceleration/deceleration is automatically applied when the tool starts and ends its movement (Fig. 5.5.1 (a)).

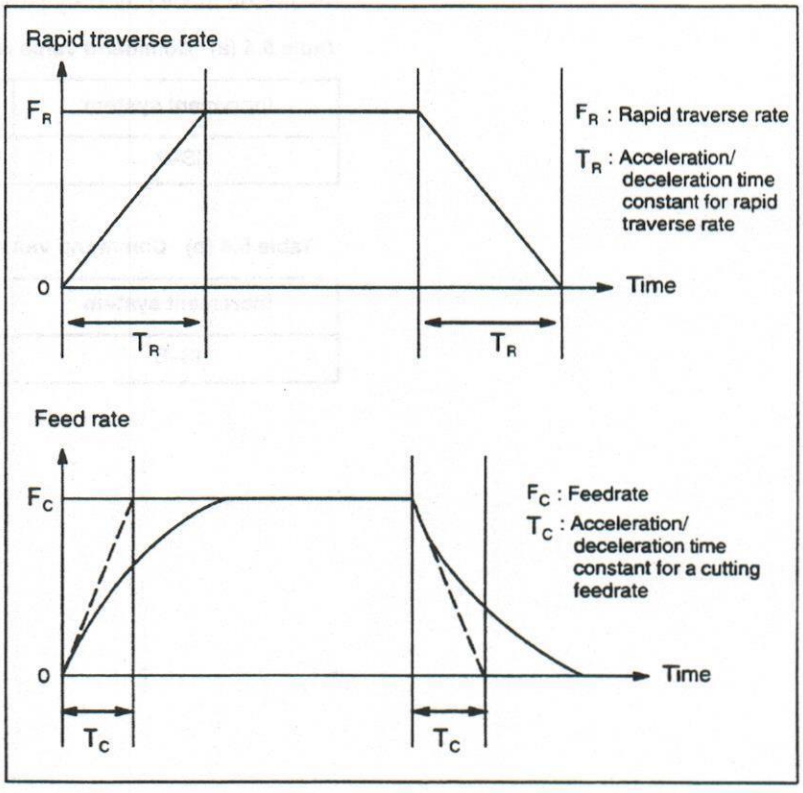


Fig. 5.5.1 (a) Automatic acceleration/deceleration (example)

Acceleration and deceleration is performed when starting and ending movement, resulting in smooth start and stop.
Automatic acceleration/deceleration is also performed when feedrate changes, so change in speed is also smoothly done.
It is not necessary to take acceleration/deceleration into consideration when programming.

Rapid traverse: Linear acceleration/deceleration (time constant per axis is set by parameter 1620)

- **Tool path in a cutting feed**

If the direction of movement changes between specified blocks during cutting feed, a rounded–corner path may result (Fig. 5.5.1 (b)).

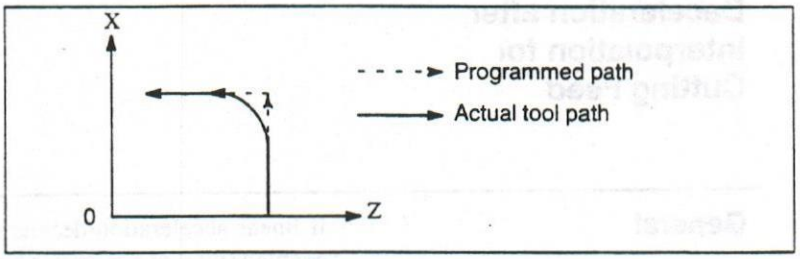


Fig. 5.5.1 (b) Example of Tool Path between Two Blocks

In circular interpolation, a radial error occurs (Fig. 5.5.1 (c)).

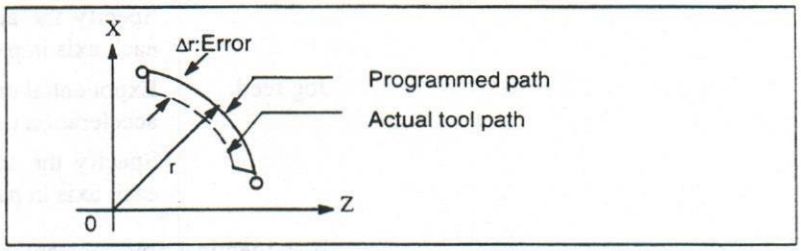


Fig. 5.5.1 (c) Example of Radial Error in Circular Interpolation

The rounded–corner path shown in Fig. 5.5.1 (b) and the error shown in Fig. 5.5.1 (c) depend on the feedrate. So, the feedrate needs to be controlled for the tool to move as programmed.

5.5.2 Linear Acceleration/ Deceleration after Interpolation for Cutting Feed

General

If linear acceleration/deceleration after interpolation for cutting feed is enabled (bit 0 of parameter No. 1610, CTL), acceleration/deceleration is performed as follows:

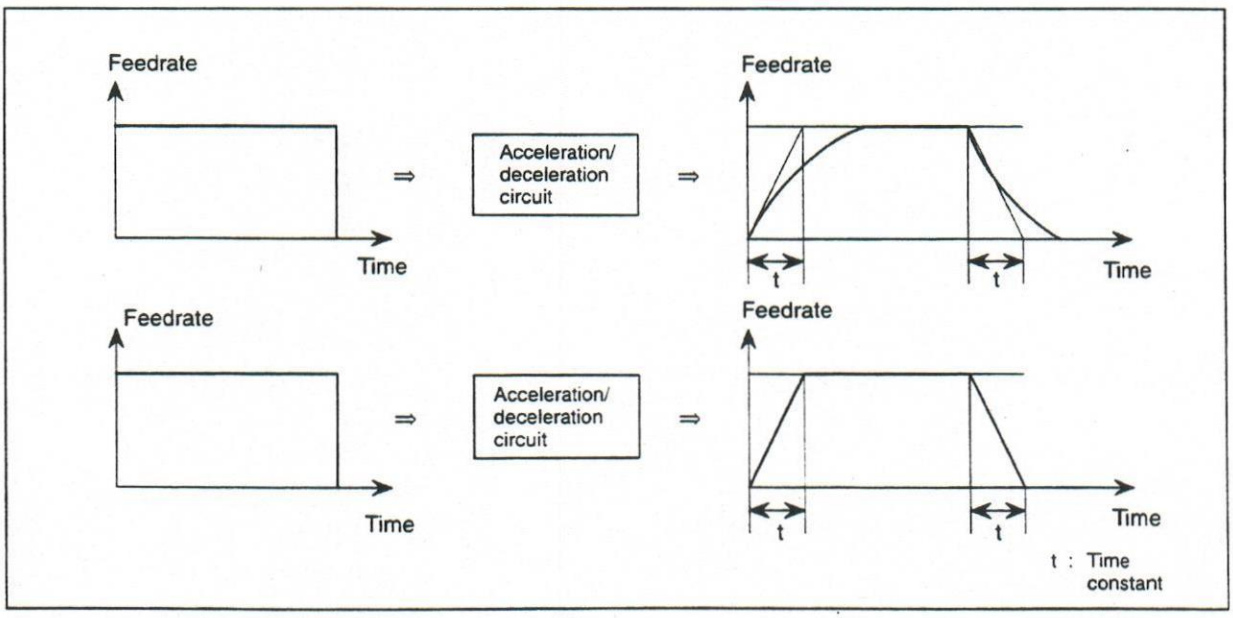
Cutting feed: Linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1622.

Jog feed: Exponential or linear acceleration/deceleration (constant acceleration time)

Specify the acceleration/deceleration time constant for each axis in parameter No. 1624.

If an identical time constant is specified, linear acceleration/deceleration can halve the delay relative to the programmed time, in comparison with exponential acceleration/deceleration, thus reducing the time needed for acceleration and deceleration. If circular interpolation is performed, especially when high–speed cutting is being performed, the actual tool path created after acceleration/deceleration will deviate from the programmed arc in the radial direction. This deviation can also be reduced, in comparison with exponential acceleration/deceleration, by applying linear acceleration/deceleration.

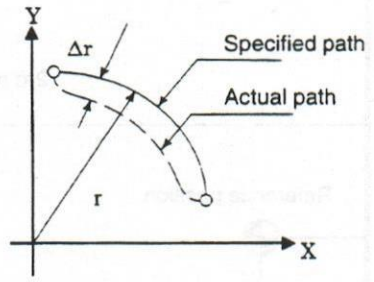


Linear acceleration/deceleration after interpolation for cutting feed is an optional function. The time constants for cutting feed and jog feed for each axis are specified in parameter Nos. 1622 and 1624 respectively, in the same way as for exponential acceleration/deceleration. The values specified for the FL feedrate for cutting feed (parameter No. 1623) and the FL feedrate for jog feed (parameter No. 1625) are ignored (always assumed to be 0).

Note

If the optional function for linear acceleration/deceleration after interpolation for cutting feed is not provided, exponential acceleration/deceleration is always selected, irrespective of the setting.

- If linear acceleration/deceleration after interpolation for cutting feed is enabled, linear acceleration/deceleration is executed during cutting feed and during a dry run. Linear acceleration/deceleration can also be executed during jog feed if the JGL (bit 4 of parameter No. 1610) is specified accordingly.
- In circular interpolation especially when circular cutting is executed at high speed, the actual path of the accelerated or decelerated tool deviates from the specified arc in the direction of the radius.



Δr : Maximum radius error (mm)
v : Feedrate (mm/s)
r : Radius of arc (mm)
T_1 : Acceleration/deceleration time constant (s)
T_2 : Time constant of servo loop (s)
(Reciprocal of position loop gain)

The maximum error in the radial direction (Δr) can be approximated by the following expressions:

$$\Delta r = \left(\frac{1}{2}T_1^2+\frac{1}{2}T_2^2\right)\frac{v^2}{r} \quad \cdots\cdots\cdots\cdots \text{Exponential acceleration/deceleration}$$

$$\Delta r = \left(\frac{1}{24}T_1^2+\frac{1}{2}T_2^2\right)\frac{v^2}{r} \quad \cdots\cdots\cdots\cdots \text{Linear acceleration/deceleration after interpolation}$$

If the error caused by the time constant of the servo loop is excluded, the error cause by linear acceleration/deceleration after interpolation is 1/12 of that caused by exponential acceleration/deceleration.

6 REFERENCE POSITION

General

• Reference position

The reference position is a fixed position on a machine tool to which the tool can easily be moved by the reference position return function.
For example, the reference position is used as a position at which tools are automatically changed. Up to two reference positions can be specified by setting coordinates in the machine coordinate system in parameters (No. 1240, No. 1241).

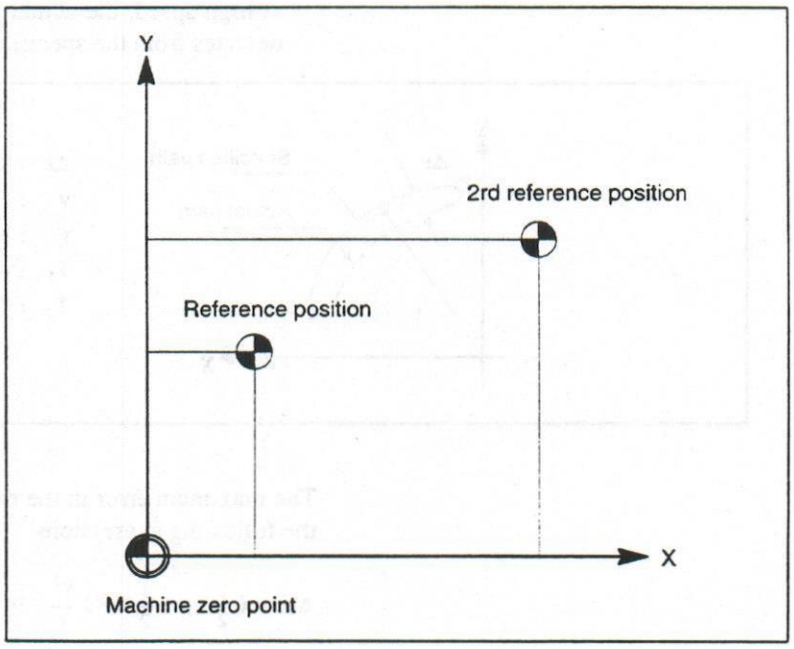


Fig. 6 (a) Machine zero point and reference positions

- **Reference position return**

Tools are automatically moved to the reference position via an intermediate position along a specified axis. When reference position return is completed, the lamp for indicating the completion of return goes on.

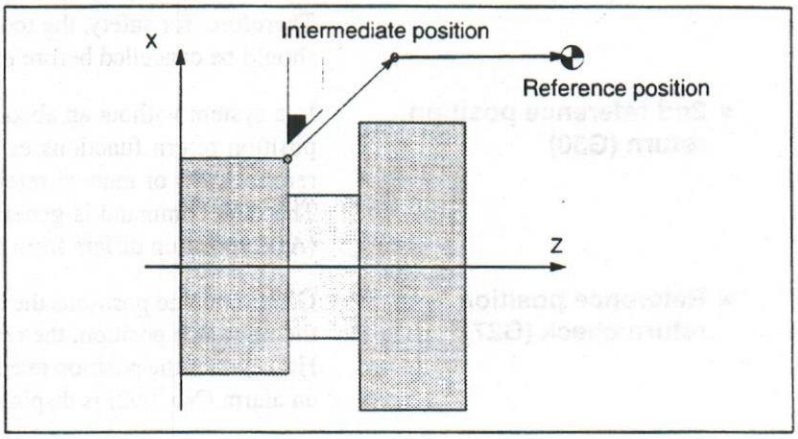


Fig. 6 (b) Reference position return

- **Reference position return check**

The reference position return check (G27) is the function which checks whether the tool has correctly returned to the reference position as specified in the program. If the tool has correctly returned to the reference position along a specified axis, the lamp for the axis goes on.

Format

- **Reference position return**

G28 IP_ ; Reference position return

G30 P2 IP_ ;2nd reference position return (P2 can be omitted.)

IP : Command specifying the intermediate position (Absolute/incremental command)

- **Reference position return check**

G27 IP_ ;

IP : Command specifying the reference position (Absolute/incremental command)

Explanations

- **Reference position return (G28)**

  Positioning to the intermediate or reference positions are performed at the rapid traverse rate of each axis.
  Therefore, for safety, the tool nose radius compensation, and tool offset should be cancelled before executing this command.

- **2nd reference position return (G30)**

  In a system without an absolute–position detector, the second reference position return functions can be used only after the reference position return (G28) or manual reference position return (see III–3.1) is made. The G30 command is generally used when the automatic tool changer (ATC) position differs from the reference position.

- **Reference position return check (G27)**

  G27 command positions the tool at rapid traverse rate. If the tool reaches the reference position, the reference position return lamp lights up.
  However, if the position reached by the tool is not the reference position, an alarm (No. 092) is displayed.

Restrictions

- **Status the machine lock being turned on**

  The lamp for indicating the completion of return does not go on when the machine lock is turned on, even when the tool has automatically returned to the reference position. In this case, it is not checked whether the tool has returned to the reference position even when a G27 command is specified.

- **First return to the reference position after the power has been turned on (without an absolute position detector)**

  When the G28 command is specified when manual return to the reference position has not been performed after the power has been turned on, the movement from the intermediate point is the same as in manual return to the reference position.
  In this case, the tool moves in the direction for reference position return specified in parameter ZMIx (bit 5 of No. 1006). Therefore the specified intermediate position must be a position to which reference position return is possible.

- **Reference position return check in an offset mode**

  In an offset mode, the position to be reached by the tool with the G27 command is the position obtained by adding the offset value. Therefore, if the position with the offset value added is not the reference position, the lamp does not light up, but an alarm is displayed instead. Usually, cancel offsets before G27 is commanded.

- **Lighting the lamp when the programmed position does not coincide with the reference position**

  When the machine tool is an inch system with metric input, the reference position return lamp may also light up even if the programmed position is shifted from the reference position by 1μ. This is because the least input increment of the machine is smaller than its least command increment.

Reference

- **Manual reference position return**

  See III–3.1.

7 COORDINATE SYSTEM

By teaching the CNC a desired tool position, the tool can be moved to the position. Such a tool position is represented by coordinates in a coordinate system. Coordinates are specified using program axes.
When two program axes, the X–axis and Z–axis, are used, coordinates are specified as follows:

X_Z_

This command is referred to as a dimension word.

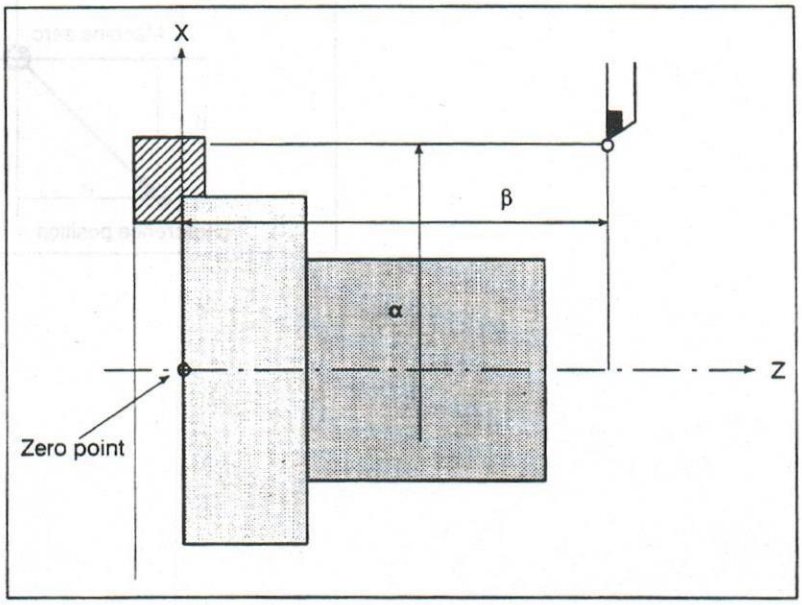


Fig. 7 Tool Position Specified by XαZβ

There are following two coordinate systems:

(1) Machine coordinate system
(2) Workpiece coordinate system

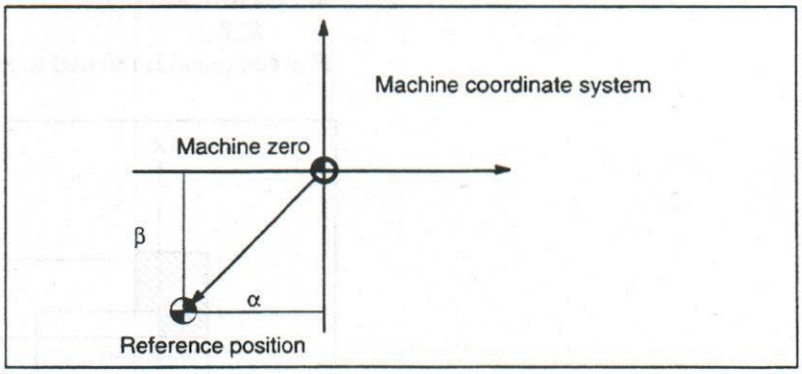
The number of the axes of a coordinate system varies from one machine to another. So, in this manual, a dimension word is represented as IP_.

7.1 MACHINE COORDINATE SYSTEM

The point that is specific to a machine and serves as the reference of the machine is referred to as the machine zero point. A machine tool builder sets a machine zero point for each machine.
A coordinate system with a machine zero point set as its origin is referred to as a machine coordinate system.
A machine coordinate system is set by performing manual reference position return after power–on (see III–3.1). A machine coordinate system, once set, remains unchanged until the power is turned off.

Explanations

When manual reference position return is performed after power–on, a machine coordinate system is set so that the reference position is at the coordinate values of (α, β) set using parameter No.1240.



7.2 WORKPIECE COORDINATE SYSTEM

A coordinate system used for machining a workpiece is referred to as a workpiece coordinate system. A workpiece coordinate system is to be set with the NC beforehand (**setting a workpiece coordinate system**).
A machining program sets a workpiece coordinate system.
A set workpiece coordinate system can be changed by shifting its origin (**Shifting a workpiece coordinate system**).

A workpiece coordinate system can be set using one of two methods:

(1) **Method using G50**
A workpiece coordinate system is set by specifying a value after G50 in the program.

(2) **Method using automatic coordinate setting**
If bit 0 of parameter No.1201 is set beforehand, a workpiece coordinate system is automatically set when manual reference position return is performed.

7.2.1 Setting a Workpiece Coordinate System (Method Using G50)

Format

Setting a workpiece coordinate system by G50

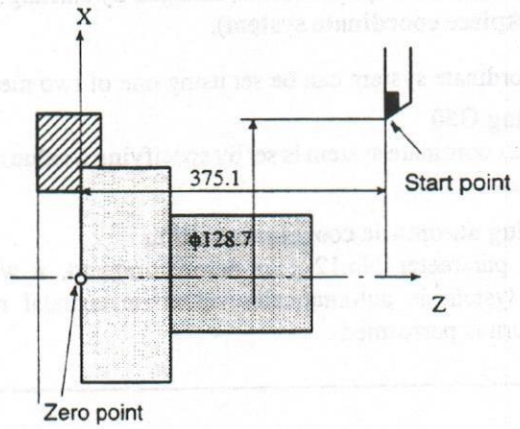
```
G50 IP_
```

Explanations

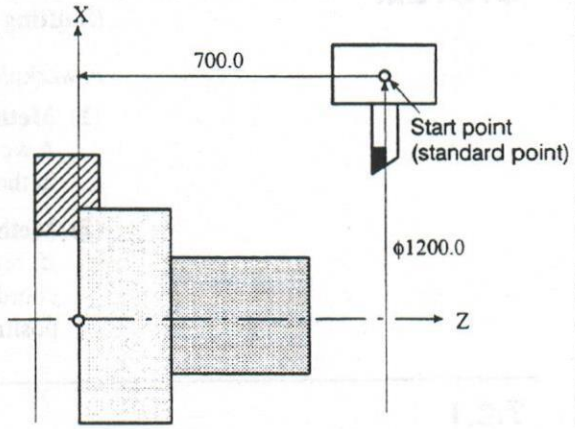
A workpiece coordinate system is set so that a point on the tool, such as the tool tip, is at specified coordinates. If IP is an incremental command value, the work coordinate system is defined so that the current tool position coincides with the result of adding the specified incremental value to the coordinates of the previous tool position. If a coordinate system is set using G50 during offset, a coordinate system in which the position before offset matches the position specified in G50 is set.

Examples

Example 1
Setting the coordinate system by the G50X128.7Z375.1; command (Diameter designation)



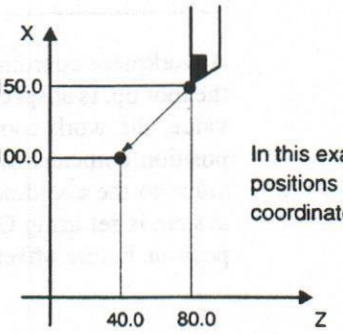
Example 2 Base point
Setting the coordinate system by the G50X1200.0Z700.0; command (Diameter designation)



Selecting a workpiece coordinate system set by G50 or automatic workpiece coordinate system setting (see Section 7.2.2)

Once a workpiece coordinate system is selected, absolute commands work with the workpiece coordinate system.

G50 X150.0 Z80.0 ;
G00 X100.0 Z40.0 ;



In this example, positioning is made to positions (X=100.0, Z=40.0) in workpiece coordinate system.

Fig. 7.2.1 (a)

7.2.2 Automatic Coordinate System Setting

When parameter ZPR (No.1201#0) for automatic coordinate system setting is set in advance, the coordinate system is determined automatically at the time of manual reference point return.
If α is set at parameter number 1250 (X axis) and β at parameter number 1250 (Z axis), a work coordinate system is set so that the standard point on the turret or the nose of the standard tool (any specific standard point) is at (X=α, Z=β) when the reference point return is over.
This is the same as when the following command is designated at the reference point.

G50X α Z β ;

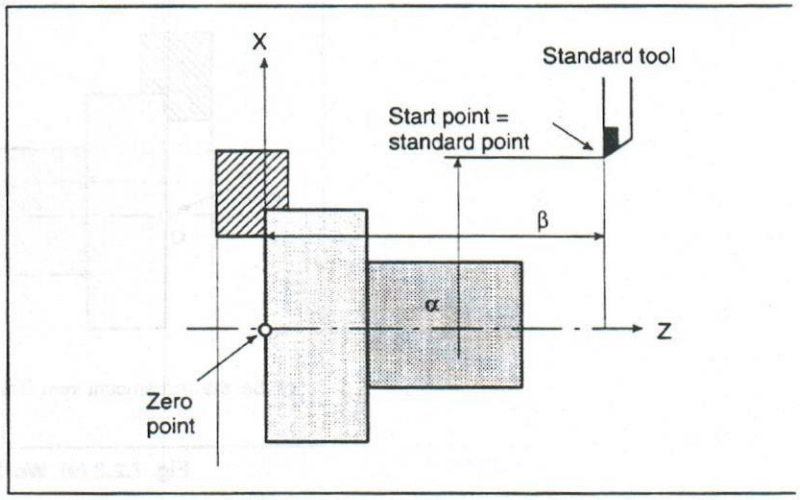


Fig. 7.2.2(a) When the tool tip of the standard tool is set as the standard point

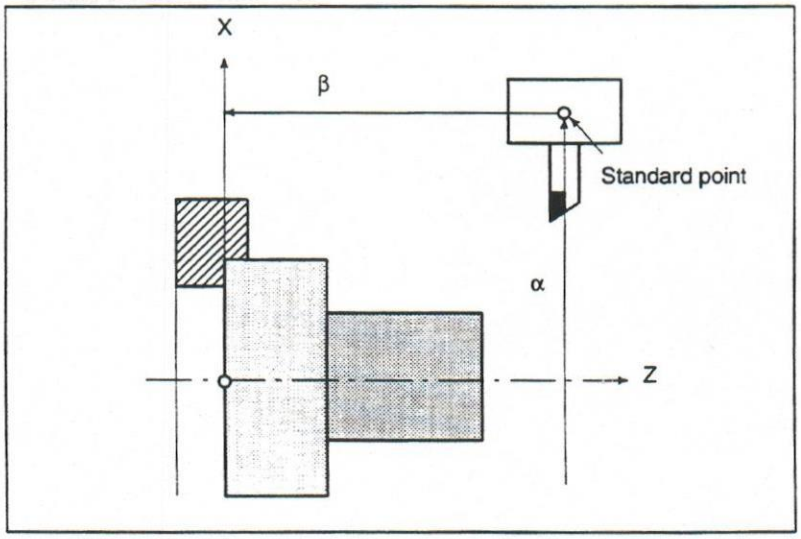


Fig 7.2.2 (b) When the turret center is set as the standard point

7.2.3 Workpiece Coordinate System Shift

Explanations

When the coordinate system actually set by the G50 command or the automatic system setting deviates from the programmed work system, the set coordinate system can be shifted.
Set the desired shift amount in the work coordinate system shift memory.

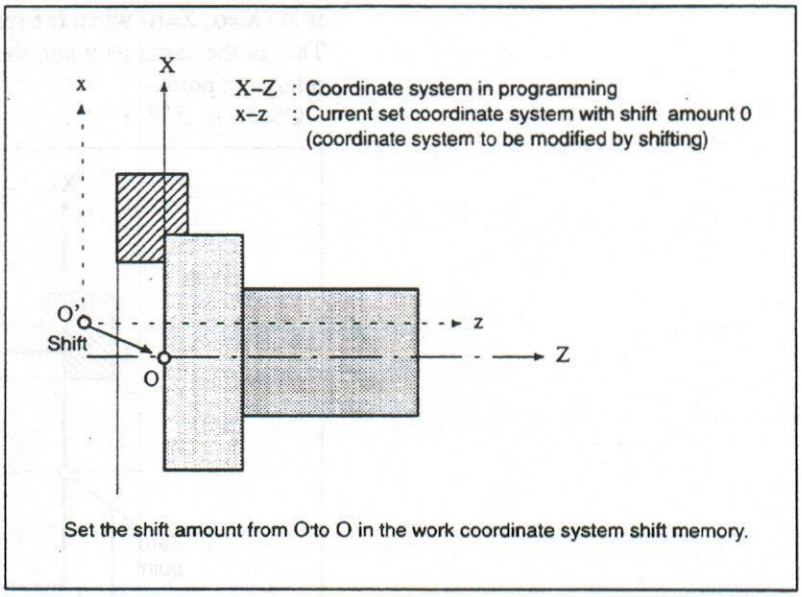

X
x
X–Z : Coordinate system in programming
x–z : Current set coordinate system with shift amount 0
(coordinate system to be modified by shifting)
O'
Shift
O
z
Z
Set the shift amount from O'to O in the work coordinate system shift memory.

Fig. 7.2.3 (a) Workpiece Coordinate System shift

See Section 11.4.4 of Part III for how to specify the distance the work coordinate system is shifted.

8 COORDINATE VALUE AND DIMENSION

This chapter contains the following topics.

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)
8.2 INCH/METRIC CONVERSION (G20, G21)
8.3 DECIMAL POINT PROGRAMMING
8.4 DIAMETER AND RADIUS PROGRAMMING

8.1 ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91)

There are two ways to command travels of the tool; the absolute command, and the incremental command. In the absolute command, coordinate value of the end position is programmed; in the incremental command, move distance of the position itself is programmed. G90 and G91 are used to command absolute or incremental command, respectively.

Absolute programming or incremental programming is used depending on the command used. See following tables.

G code system	A	B or C
Command method	Address word	G90, G91

Format

- **G code system A**

	Absolute command	Incremental command
X axis move command Z axis move command	X Z	U W

- **G code system B or C**

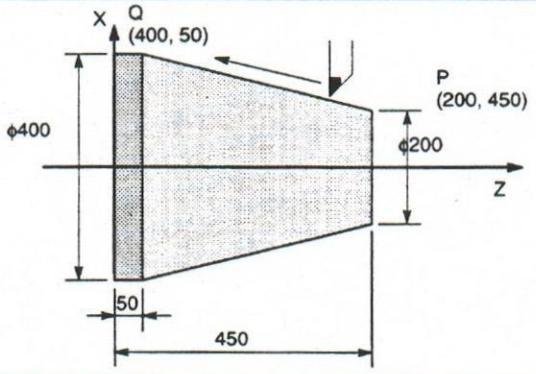
Absolute command G90 IP_ ;

Incremental command G91 IP_ ;

Examples

- **Tool movement from point P to point Q (diameter programming is used for the X–axis)**

	G code system A	G code system B or C
Absolute command	X400.0 Z50.0 ;	G90 X400.0 Z50.0 ;
Incremental command	U200.0 W–400.0 ;	G91 X200.0 Z–400.0 ;



Notes

1. Absolute and incremental commands can be used together in a block.
   In the above example, the following command can be specified :
   X400.0 W–400.0 ;
2. When both X and U or W and Z are used together in a block, the one specified later is effective.

8.2 INCH/METRIC CONVERSION (G20, G21)

Either inch or metric input can be selected by G code.

Format

G20 ;	Inch input
G21 ;	mm input

This G code must be specified in an independent block before setting the coordinate system at the beginning of the program. After the G code for inch/metric conversion is specified, the unit of input data is switched to the least inch or metric input increment of increment system IS–B (Section 2.3). The unit systems for the following values are changed after inch/metric conversion:

- **Feedrate commanded by F code**
- **Positional command**
- **Workpiece coordinate sytem shift value**
- **Tool compensation value**
- **Unit of scale for manual pulse generator**
- **Movement distance in incremental feed**
- **Some parameters**

When the power is turned on, the G code is the same as that held before the power was turned off.

Notes

1. G20 and G21 must not be switched during a program.
2. When the least input increment and the least command increment systems are different, the maximum error is half of the least command increment. This error is not accumulated.
3. When switching inch input (G20) to metric input (G21) and vice versa, the tool compensation value must be re–set according to the least input increment.
   However, when bit 0 (OIM) of parameter 5006 is 1, tool compensation values are automatically converted and need not be re–set.
4. When the G28 command is specified first after inch input is switched to metric input or vice versa, movement from the intermediate point is the same as for manual return to the reference position. In this case, the tool moves in the direction for reference position return, as specified with bit 5 (ZMI) of parameter No.1006.
5. The inch and metric input can also be switched using setting the Setting Data (see III–11.4.5).

8.3 DECIMAL POINT PROGRAMMING

Numerical values can be entered with a decimal point. A decimal point can be used when entering a distance, time, or speed. Decimal points can be specified with the following addresses:
X, Z, U, W, C, I, K, R, and F.

Explanations

There are two types of decimal point notation: calculator–type notation and standard notation.
When calculator–type decimal notation is used, a value without decimal point is considered to be specified in mm, inch, or min. When standard decimal notation is used, such a value is considered to be specified in least input increments. Select either calculator–type or standard decimal notation by using the DPI bit (bit 0 of parameter 3401).Values can be specified both with and without decimal point in a single program.

Examples

Program command	Pocket calculator type decimal point programming	Standard type decimal point programming
X1000 Command value without decimal point	1000mm Unit : mm	1mm Unit : Least input increment (0.001 mm)
X1000.0 Command value with decimal point	1000mm Unit : mm	1000mm Unit : mm

Notes

1. In a single block, specify a G code before entering a value. The position of decimal point may depend on the command.

   Examples:

   G20; Input in inches

   X1.0 G04; X1.0 is considered to be a distance and processed as X10000. This command is equivalent to G04 X10000. The tool dwells for 10 seconds.

   G04 X1.0; Equivalent to G04 X1000. The tool dwells for one second.

2. Fractions less than the least input increment are truncated.

   Examples:

   X1.2345; Truncated to X1.234 when the least input increment is 0.001 mm.
   Processed as X1.2345 when the least input increment is 0.0001 inch.

3. When more than eight digits are specified, an alarm occurs. If a value is entered with a decimal point, the number of digits is also checked after the value is converted to an integer according to the least input increment.

   Examples:

   X1.23456789; Alarm 003 occurs because more than eight digits are specified.

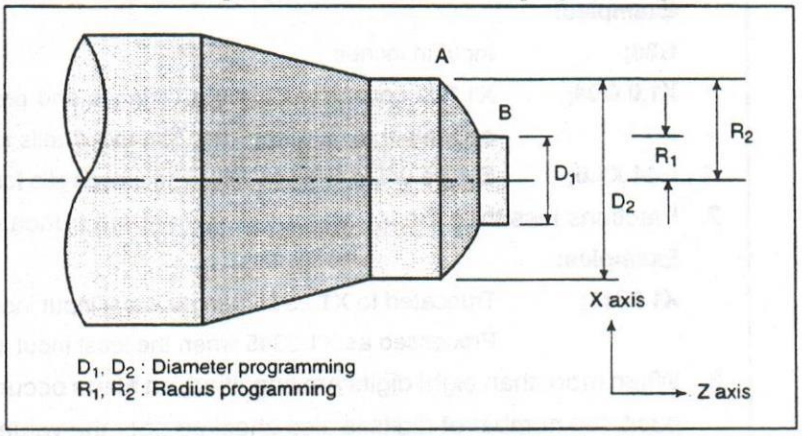
   X123456.7; If the least input increment is 0.001 mm, the value is converted to integer 123456700. Because the integer has more than eight digits, an alarm occurs.

8.4 DIAMETER AND RADIUS PROGRAMMING

Since the work cross section is usually circular in CNC lathe control programming, its dimensions can be specified in two ways :

Diameter and Radius

When the diameter is specified, it is called diameter programming and when the radius is specified, it is called radius programming.



Explanations

• Notes on diameter programming/radius programming for each command

Radius programming or diameter programming can be specified by parameter DIA (No.1006#3). When using diameter programming, note the conditions listed in the table 8.4(a).

Table 8.4(a) Notes on specifying diameter value

Item	Notes
X axis command	Specified with a diameter value
Incremental command	Specified with a diameter value In the above figure, specifies D2 minus D1 for tool path B to A.
Coordinate system setting (G50)	Specifies a coordinate value with a diameter value
Component of tool offset value	Parameter (No.5004#1) determines either diameter or radius value
Parameters in canned cycle, such as cutting depth along X axis. (R)	Specifies a radius value
Radius designation in circular interpolation (R, I, K, and etc.)	Specifies a radius value
Feedrate along axis	Specifies change of radius/rev. or change of radius/min.
Display of axis position	Displayed as diameter value

9 SPINDLE SPEED FUNCTION

The spindle speed can be controlled by specifying a value following address S.
This chapter contains the following topics.

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE
9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND)
9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

9.1 SPECIFYING THE SPINDLE SPEED WITH A CODE

By specifying a numerical value following address S, a binary code signal and a strobe signal are transmitted to the machine tool. This is mainly used to control the spindle speed. A block can contain only one S code. Refer to the appropriate manual provided by the machine tool builder for details such as the number of digits in an S code or the execution order when a move command and an S code command are in the same block.

9.2 SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND)

The spindle speed can be specified directly by address S followed by a five–digit value (rpm). The unit for specifying the spindle speed may vary depending on the machine tool builder. Refer to the appropriate manual provided by the machine tool builder for details.

9.3 CONSTANT SURFACE SPEED CONTROL (G96, G97)

Specify the surface speed (relative speed between the tool and workpiece) following S. The spindle is rotated so that the surface speed is constant regardless of the position of the tool.

Format

- **Constant surface speed control command**

```
G96 S○○○○○ ;
      ↑Surface speed (m/min or feet/min)

Note : This surface speed unit may change according to
       machine tool builder's specification.
```

- **Constant surface speed control cancel command**

```
G97 S○○○○○ ;
      ↑Spindle speed (rpm)

Note : This surface speed unit may change according to
       machine tool builder's specification.
```

- **Clamp of maximum spindle speed**

```
G50 S_ ;    The maximum spindle speed (rpm) follows S.
```

Explanations

- **Constant surface speed control command (G96)**

G96 (constant surface speed control command) is a modal G code. After a G96 command is specified, the program enters the constant surface speed control mode (G96 mode) and specified S values are assumed as a surface speed. A G97 command cancels the G96 mode.When constant surface speed control is applied, a spindle speed higher than the value specified in G50S_; (maximum spindle speed) is clamped at the maximum spindle speed. When the power is turned on, the maximum spindle speed is not yet set and the speed is not clamped.S (surface speed) commands in the G96 mode are assumed as S = 0 (the surface speed is 0) until M03 (rotating the spindle in the positive direction) or M04 (rotating the spindle in the negative direction) appears in the program.

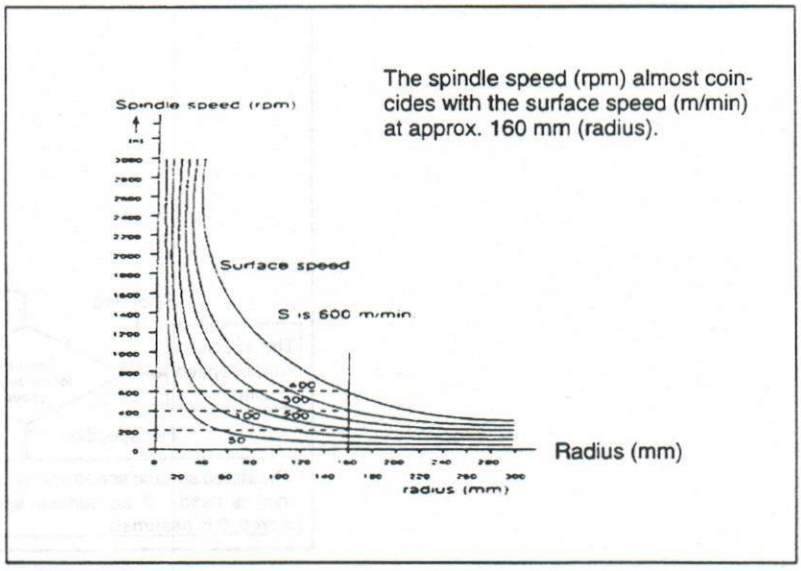

The spindle speed (rpm) almost coincides with the surface speed (m/min) at approx. 160 mm (radius).
Spindle speed (rpm)
Surface speed
S is 600 m/min
Radius (mm)
radius (mm)

Fig. 9.3 (a) Relation between workpiece radius, spindle speed and surface speed

- **Setting the workpiece coordinate system for constant surface speed control**

To execute the constant surface speed control, it is necessary to set the work coordinate system , Z axis, (axis to which the constant surface speed control applies) becomes zero.

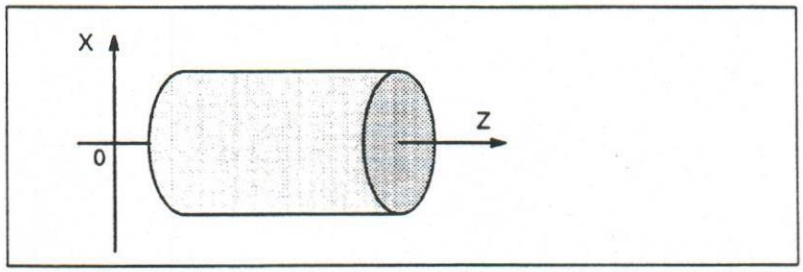
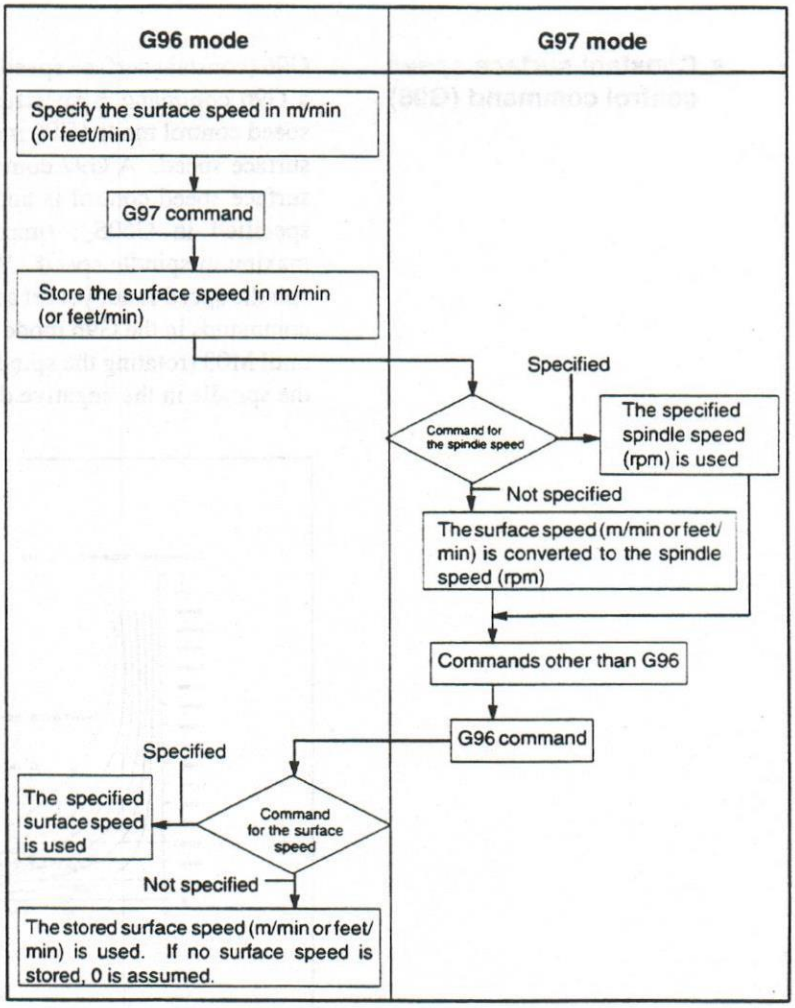


Fig. 9.3 (b) Example of the Workpiece Coordinate System for Constant Surface Speed Control

- **Surface speed specified in the G96 mode**


G96 mode
G97 mode
Specify the surface speed in m/min (or feet/min)
G97 command
Store the surface speed in m/min (or feet/min)
Specified
Command for the spindle speed
The specified spindle speed (rpm) is used
Not specified
The surface speed (m/min or feet/min) is converted to the spindle speed (rpm)
Commands other than G96
G96 command
Specified
The specified surface speed is used
Command for the surface speed
Not specified
The stored surface speed (m/min or feet/min) is used. If no surface speed is stored, 0 is assumed.

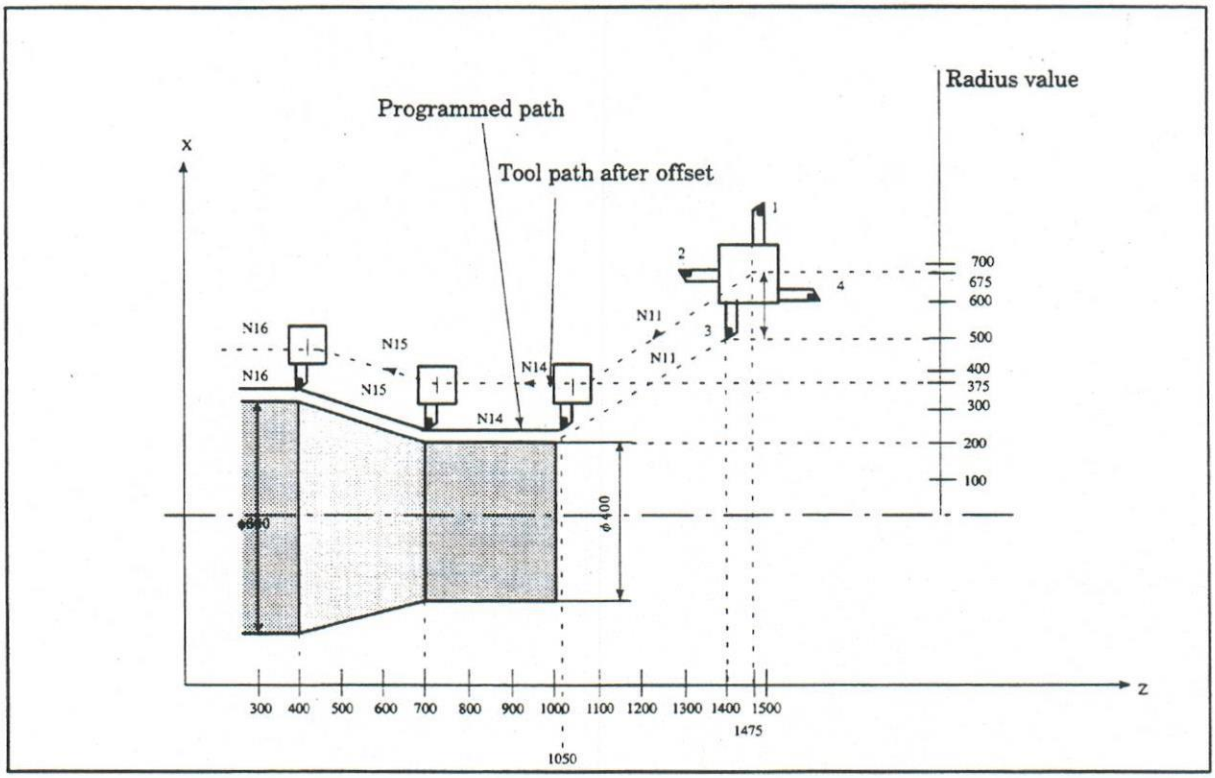
Restrictions

- **Constant surface speed control for threading**

The constant surface speed control is also effective during threading. Accordingly, it is recommended that the constant surface speed control be invalidated with G97 command before starting the scroll threading and taper threading, because the response problem in the servo system may not be considered when the spindle speed changes.

- **Constant surface speed control for rapid traverse (G00)**

In a rapid traverse block specified by G00, the constant surface speed control is not made by calculating the surface speed to a transient change of the tool position, but is made by calculating the surface speed based on the position at the end point of the rapid traverse block, on the condition that cutting is not executed at rapid traverse.

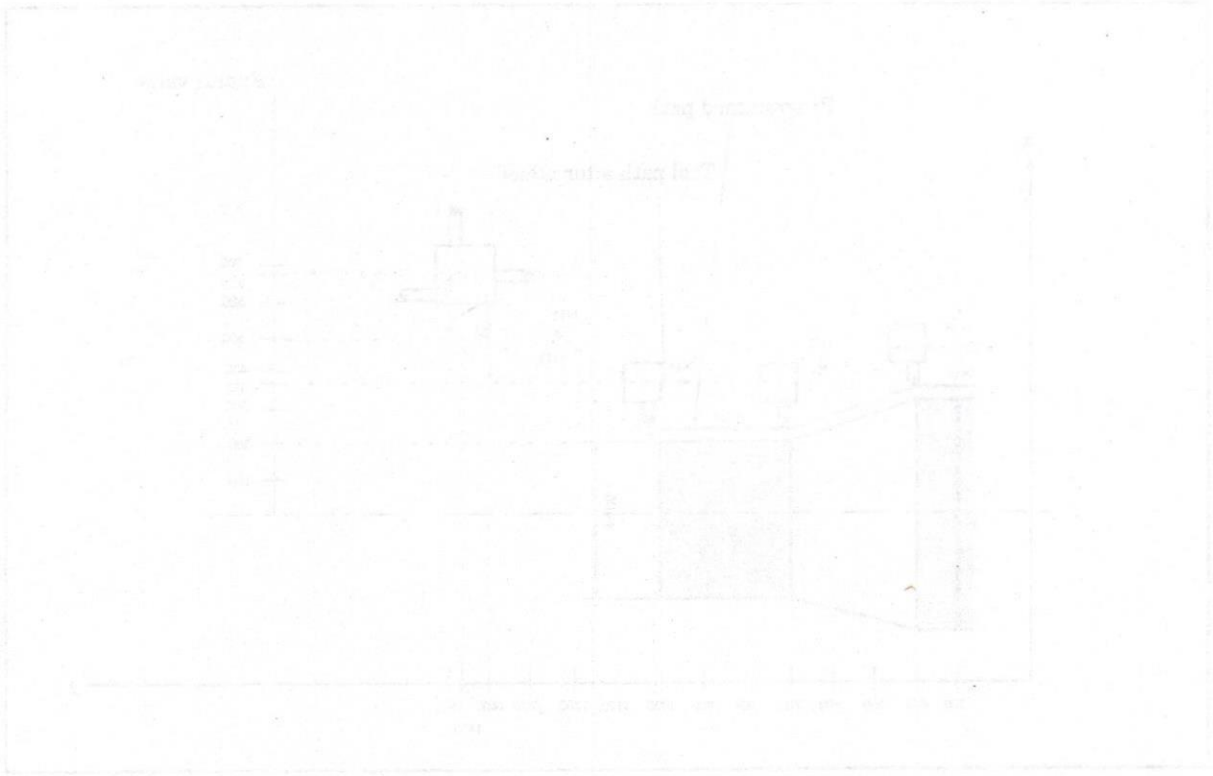


Example

```
N8  G00 X1000.0Z1400.0 ;
N9  T33;
N11 X400.0Z1050.0;
N12 G50S3000 ; (Designation of max. spindle speed)
N13 G96S200 ; (Surface speed 200m/min)
N14 G01 Z 700.0F1000 ;
N15 X600.0Z 400.0;
N16 Z ... ;
```

The CNC calculates the spindle speed which is proportional to the specified surface speed at the position of the programmed coordinate value on the X axis. This is not the value calculated according to the X axis coordinate after offset when offset is valid. At the end point N15 in the example above, the speed at 600 dia. (Which is not the turret center but the tool nose) is 200 m/min. If X axis coordinate value is negative, the CNC uses the absolute value.

10 TOOL FUNCTION (T FUNCTION)



10.1 TOOL SELECTION

By specifying a 2–digit/4–digit numerical value following address T, a code signal and a strobe signal are transmitted to the machine tool. This is mainly used to select tools on the machine.
One T code can be commanded in a block. Refer to the machine tool builder's manual for the number of digits commandable with address T and the correspondence between the T codes and machine operations.
When a move command and a T code are specified in the same block, the commands are executed in one of the following two ways:

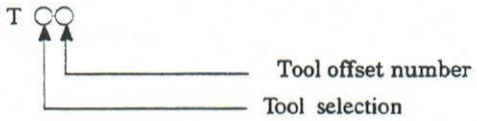
1. Simultaneous execution of the move command and T function commands.
2. Executing T function commands upon completion of move command execution.

The selection of either sequence depends on the machine tool builder's specifications. Refer to the machine tool builder's manual for details.

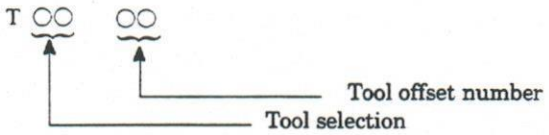
Explanations

The value after the T code indicates the desired tool. Part of the values is also used as the offset number indicating the compensation amount for tool offset.
The T code has the following two formats. The format to be used can be selected by bit 0 (LD1) of parameter No.5002.

1.Last one digit of T–code designates the offset number.



2.Last two digits of T–code designate the offset number.



Refer to the machine tool builder's manual for correspondence between the T–code and the tool and the number of digit to specify tool selection.
Example(T2+2)
N1G00X100.0Z140.0
N2T0313; . . . . . (Select Tool No. 3 and Offset value No.13)
N3X40.0Z105.0;
Some machines use a 1–digit value to specify tool selection.

11 AUXILIARY FUNCTION

General

There is auxiliary function (M code) for specifying spindle start, spindle stop program end, and so on.

When a move command and miscellaneous function are specified in the same block, the commands are executed in one of the following two ways:

i) Simultaneous execution of the move command and miscellaneous function commands.

ii) Executing miscellaneous function commands upon completion of move command execution.

The selection of either sequence depends on the machine tool builder's specification. Refer to the manual issued by the machine tool builder for details.

11.1 AUXILIARY FUNCTION (M FUNCTION)

When address M followed by a number is specified, a code signal and strobe signal are transmitted. These signals are used for turning on/off the power to the machine.
In general, only one M code is valid in a block but up to three M codes can be specified in a block (although some machines may not allow that). The correspondence between M codes and functions is up to the machine tool builder.
All M codes are processed in the machine except for M98, M99, M codes for calling a subprogram (parameters Nos. 6071 to 6079), and M codes for calling a custom macro (parameters Nos. 6080 to 6089). Refer to the appropriate manual issued by the machine tool builder.

Explanations

The following M codes have special meanings.

- **M02, M03 (End of program)**

  This indicates the end of the main program
  Automatic operation is stopped and the CNC unit is reset. This differs with the machine tool builder. After a block specifying the end of the program is executed,control returns to the start of the program. Bit 4, 5 of parameter 3404 (M30, M02) can be used to disable from returning control to the start of the program.

- **M00 (Program stop)**

  Automatic operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged. The automatic operation can be restarted by actuating the cycle operation. This differs with the machine tool builder.

- **M01 (Optional stop)**

  Similarly to M00, automatic operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel has been pressed.

- **M98 (Calling of sub-program)**

  This code is used to call a subprogram. The code and strobe signals are not sent. See the subprogram **section 12.3** for details .

- **M99 (End of subprogram)**

  This code indicates the end of a subprogram.
  M99 execution returns control to the main program. The code and strobe signals are not sent. See the subprogram **section 12.3** for details.

Notes

The block following M00, M01, M02 and M30, is not read into the input buffer register, if present. Similarly, ten M codes which do not buffer can be set by parameters (Nos. 3411 to 3420). Refer to the machine tool builder's instruction manual for these M codes.

11.2 MULTIPLE M COMMANDS IN A SINGLE BLOCK

So far, one block has been able to contain only one M code. However, by specifying parameter M3B (No.3404#7) to 1, this function allows up to three M codes to be contained in one block.
Up to three M codes specified in a block are simultaneously output to the machine. This means that compared with the conventional method of a single M command in a single block, a shorter cycle time can be realized in machining.

Explanations

CNC allows up to three M codes to be specified in one block. However, some M codes cannot be specified at the same time due to mechanical operation restrictions. For example, M42 can be specified only after the mechanical operation of M41 is completed. For detailed information about the mechanical operation restrictions on simultaneous specification of multiple M codes in one block, refer to the manual of each machine tool builder.
M00, M01, M02, M30, M98, or M99 must not be specified together with another M code.
Some M codes other than M00, M01, M02, M30, M98, and M99 cannot be specified together with other M codes; each of those M codes must be specified in a single block.
Such M codes include these which direct the CNC to perform internal operations in addition to sending the M codes themselves to the machine. To be specified, such M codes are M codes for calling program numbers 9001 to 9009 and M codes for disabling advance reading (buffering) of subsequent blocks. Meanwhile, multiple of M codes that direct the CNC only to send the M codes themselves (without performing internal operations) can be specified in a single block.

Examples

One M command in a single block	Multiple M commands in a single block
M40 ; M50 ; M60 ; G28G91X0Z0 ; : : :	M40M50M60 ; G28G91X0Z0 ; : : : : :

12 PROGRAM CONFIGURATION

General

• Main program and subprogram

There are two program types, main program and subprogram. Normally, the CNC operates according to the main program. However, when a command calling a subprogram is encountered in the main program, control is passed to the subprogram. When a command specifying a return to the main program is encountered in a subprogram, control is returned to the main program.

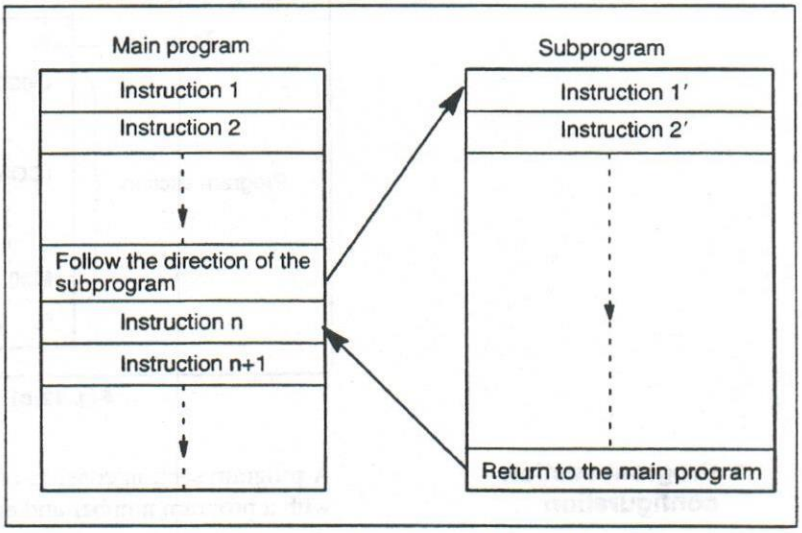


Fig. 12 (a) Main program and Subprogram

The CNC memory can hold up to 63 main programs and subprograms. A main program can be selected from the stored main programs to operate the machine. See Chapter 10 in OPERATION for the methods of registering and selecting programs.

• Program components

A program consists of the following components:

Table 12(a) Program components

Components	Descriptions
Tape start	Symbol indicating the start of a program file
Leader section	Used for the title of a program file, etc.
Program start	Symbol indicating the start of a program
Program section	Commands for machining
Comment section	Comments or directions for the operator
Tape end	Symbol indicating the end of a program file

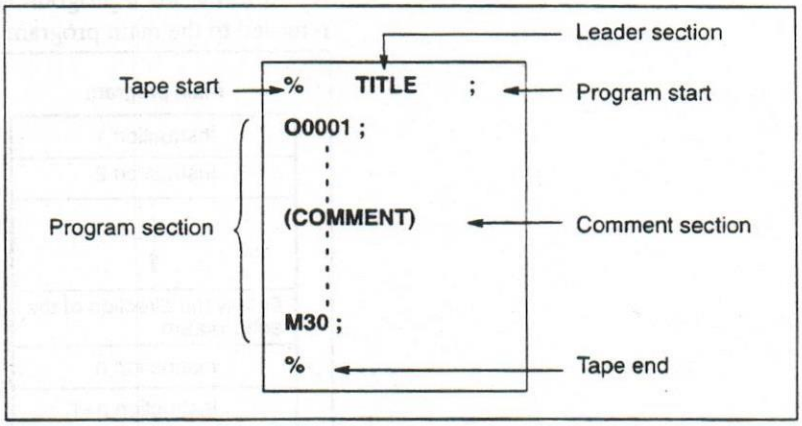


Fig. 12(b) Program configuration

• Program section configuration

A program section consists of several blocks. A program section starts with a program number and ends with a program end code.

Program section configuration	Program section
Program number	O0001 ;
Block 1	N1 G91 G00 X120.0 Y80.0 ;
Block 2	N2 G43 Z-32.0 H01 ;
: :	
Block n	Nn Z0 ;
Program end	M30 ;

A block contains information necessary for machining, such as a move command or coolant on/off command. Specifying a slash (/) at the start of a block disables the execution of some blocks (see "optional block skip" in Section 12.2).

12.1 PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS

This section describes program components other than program sections. See Section 12.2 for a program section.

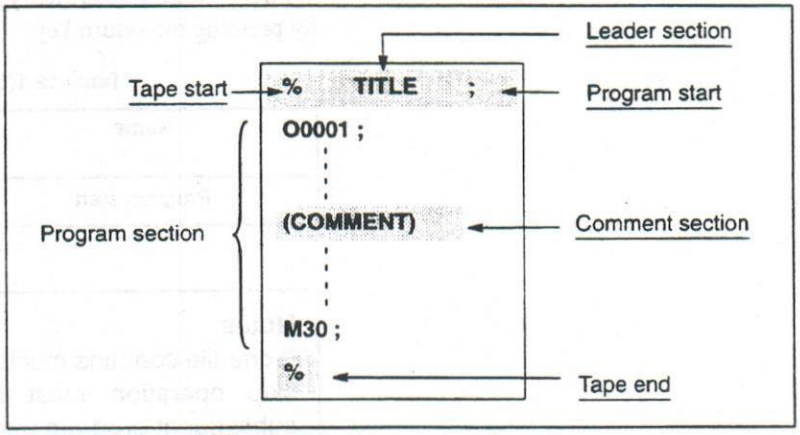


Fig. 12.1(a) Program configuration

Explanations

- **Tape start**

The tape start indicates the start of a file that contains CNC programs. The mark is not required when programs are entered using SYSTEM P or ordinary personal computers. The mark is not displayed on the CRT display screen. However, if the file is output,the mark is automatically output at the start of the file.

Table 12.1(a) Code of a tape start

Name	ISO code	EIA code	Notation in this manual
Tape start	%	ER	%

- **Leader section**

Data entered before the programs in a file constitutes a leader section. When machining is started, the label skip state is usually set by turning on the power or resetting the system. In the label skip state, all information is ignored until the first end–of–block code is read. When a file is read into the CNC unit from an I/O device, leader sections are skipped by the label skip function.
A leader section generally contains information such as a file header. When a leader section is skipped, even a TV parity check is not made. So a leader section can contain any codes except the EOB code.

• Program start

The program start code is to be entered immediately after a leader section, that is, immediately before a program section. This code indicates the start of a program, and is always required to disable the label skip function. With SYSTEM P or ordinary personal computers, this code can be entered by pressing the return key.

Table 12.1(b) Code of a program start

Name	ISO code	EIA code	Notation in this manual
Program start	LF	CR	;

Notes
If one file contains multiple programs, the EOB code for label skip operation must not appear before a second or subsequent program number.

• Comment section

Any information enclosed by the control–out and control–in codes is regarded as a comment. The user can enter a header, comments, directions to the operator, etc. in a comment section using the EOB code or any other code.

Table 12.1(c) Codes of a control–in and a control–out

Name	ISO code	EIA code	Notation in this manual	Meaning
Control–out	(	2–4–5	(	Start of comment section
Control–in	)	2–4–7	)	End of comment section

When a program is read into memory for memory operation, comment sections, if any, are not ignored but are also read into memory. Note, however, that codes other than those listed in the code table in Appendix 6 are ignored, and thus are not read into memory. When a program in this memory is output to an external device (see III–8), the comments in the program are also output.
When a program is displayed on the screen, its comment sections are also displayed. However, those codes that were ignored when read into memory are not output or displayed.

Notes
1. If a long comment section appears in the middle of a program section, a move along an axis may be suspended for a long time because of such a comment section. So a comment section should be placed where movement suspension may occur or no movement is involved.
2. If only a control–in code is read with no matching control–out code, the read control–in code is ignored.

- **Tape end**

A tape end is to be placed at the end of a file containing NC programs. If programs are entered using the automatic programming system, the mark need not be entered. The mark is not displayed on the CRT display screen. However, when a file is output, the mark is automatically output at the end of the file.
If an attempt is made to execute % when M02 or M03 is not placed at the end of the program, the alarm (No. 5010) is occurred.

Table 12.1(d) Code of a tape end

Name	ISO code	EIA code	Notation in this manual
Tape end	%	ER	%

12.2 PROGRAM SECTION CONFIGURATION

This section describes elements of a program section. See Section 12.1 for program components other than program sections.

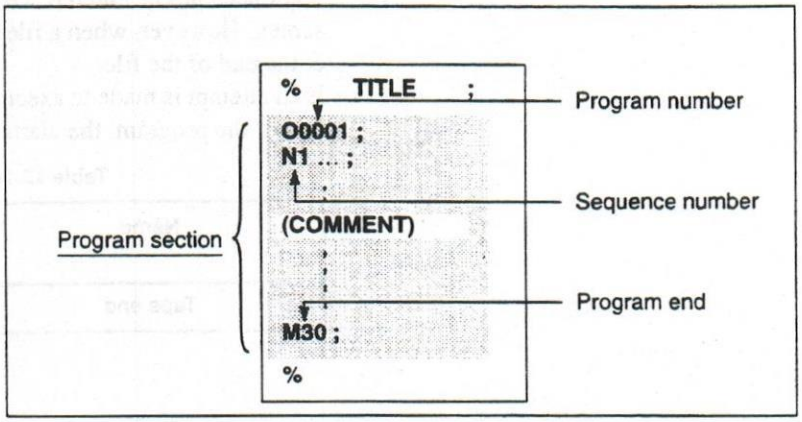


Fig. 12.2(a) Program configuration

- **Program number**

A program number consisting of address O followed by a four–digit number is assigned to each program at the beginning registered in memory to identify the program.
In ISO code, the colon (:) can be used instead of O.

Notes
Program numbers 8000 to 9999 may be used by machine tool builders, and the user may not be able to use these numbers.

• Sequence number and block

A program consists of several commands. One command unit is called a block. One block is separated from another with an end of block code (EOB).

Table 12.2(a) EOB code

Name	ISO code	EIA code	Notation in this manual
End of block (EOB)	LF	CR	;

At the head of a block, a sequence number consisting of address N followed by a number not longer than five digits (1 to 99999) can be placed. Sequence numbers can be specified in a random order, and any numbers can be skipped. Sequence numbers may be specified for all blocks or only for desired blocks of the program. In general, however, it is convenient to assign sequence numbers in ascending order in phase with the machining steps (for example, when a new tool is used by tool replacement, and machining proceeds to a new surface with table indexing.)

N300 X200.0 Z300.0 ; A sequence number is underlined.

Fig. 12.2(b) Sequence number and block (example)

Notes
N0 must not be used for the reason of file compatibility with other CNC systems.
Program number 0 cannot be used. So 0 must not be used for a sequence number regarded as a program number.

• TV check (Vertical parity check along tape)

A parity check is made for a block on input tape vertically. If the number of characters in one block (starting with the code immediately after an EOB and ending with the next EOB) is odd, an alarm (No.002) is output. No TV check is made only for those parts that are skipped by the label skip function. Whether count a comment section enclosed in parenthesis as the number of characters for TV check or not, it is selected by parameter CTV (No.0100#1). The TV check function can be enabled or disabled by setting on the MDI unit (See subsec. 11.4.5 in Part III.).

• Block configuration (word and address)

A block consists of one or more words. A word consists of an address followed by a number some digits long. (The plus sign (+) or minus sign (–) may be prefixed to a number.)

Word = Address + number (Example : X–1000)

For an address, one of the letters (A to Z) is used ; an address defines the meaning of a number that follows the address. Table 12.2 (b) indicates the usable addresses and their meanings.

The same address may have different meanings, depending on the preparatory function specification.

Table 12.2(b) Major functions and addresses

Function	Address	Meaning
Program number	O (1)	Program number
Sequence number	N	Sequence number
Preparatory function	G	Specifies a motion mode (linear, arc, etc.)
Dimension word	X, Z, U, W	Coordinate axis move command
	I, K	Coordinate of the arc center
	R	Arc radius
	I, K, R, C	chamfering, corner R value
Feed function	F	Rate of feed per minute, Rate of feed per revolution
Spindle speed function	S	Spindle speed
Tool function	T	Tool number
Auxiliary function	M	On/off control on the machine tool
Dwell	P, X, U	Dwell time
Program number designation	P	Subprogram number
Number of repetitions	P	Number of subprogram repetitions

Notes

1. In ISO code, the colon (:) can also be used as the address of a program number.

N_	G_	X_ Z_	F_	S_	T_	M_	;
Sequence number	Preparatory function	Dimension word	Feed–function	Spindle speed function	Tool function	Miscellaneous function	

Fig. 12.2 (c) 1 block (example)

• Major addresses and ranges of command values

Major addresses and the ranges of values specified for the addresses are shown below. Note that these figures represent limits on the CNC side, which are totally different from limits on the machine tool side. For example, the CNC allows a tool to traverse up to about 100 m (in millimeter input) along the X axis.
However, an actual stroke along the X axis may be limited to 2 m for a specific machine tool.
Similarly, the CNC may be able to control a cutting federate of up to 240 m/min, but the machine tool may not allow more than 3 m/min. When developing a program, the user should carefully read the manuals of the machine tool as well as this manual to be familiar with the restrictions on programming.

Table 12.2(c) Major addresses and ranges of command values

Function	Address	Input in mm	Input in inch
Program number	O (1)	1–9999	1–9999
Sequence number	N	1–99999	1–99999
Preparatory function	G	0–99	0–99
Dimension word	X, Z, U, W, C, I, K, R	–99999.999 – +99999.999	–9999.9999– +9999.9999
Feed per minute	F	1–240000mm/min	0.01–9600.00 inch/min
Feed per revolution	F	0.0001–500.0000 mm/rev	0.000001– 9.999999 inch/rev
Spindle speed function	S	0–20000	0–20000
Tool function	T	0–99999999	0–99999999
Auxiliary function	M	0–99999999	0–99999999
Dwell	P, X, U	0–99999.999s	0–99999.999s
Designation of a program number	P	1–9999	1–9999
Number of repetitions	P	1–999	1–999

Note
1. In ISO code, the colon (:) can also be used as the address of a program number.

- **Optional block skip**

When a slash (/) is specified at the head of a block, and optional block skip switch on the machine operator panel is set to on, the information contained in the block for which / is specified is ignored in DNC operation or memory operation.

When optional block skip switch is set to off, the information contained in the block for which / is specified is valid. This means that the operator can determine whether to skip the block containing /.

This function is ignored when programs are loaded into memory. Blocks containing /n are also stored in memory, regardless of how the optional block skip switch is set.

Programs held in memory can be output, regardless of how the optional block skip switches are set.

Optional block skip is effective even during sequence number search operation.

Notes

1. ***Position of a slash***
   A slash (/) must be specified at the head of a block. If a slash is placed elsewhere, the information from the slash to immediately before the EOB code is ignored.
2. ***TV and TH check***
   When an optional block skip switch is on. TH and TV checks are made for the skipped portions in the same way as when the optional block skip switch is off.
3. ***Disabling an optional block skip switch***
   Optional block skip operation is processed when blocks are read from memory or tape into a buffer. Even if a switch is set to on after blocks are read into a buffer, the blocks already read are not ignored.

- **Program end**

The end of a program is indicated by programming one of the following codes at the end of the program:

Table 12.2(d) Code of a program end

Code	Meaning usage
M02	For main program
M30	
M99	For subprogram

If one of the program end codes is executed in program execution, the CNC terminates the execution of the program, and the reset state is set. When the subprogram end code is executed, control returns to the program that called the subprogram.

Notes
A block containing an optional block skip code such as /M02 ; , /M30 ; , or /M99 ; is not regarded as the end of a program, if the optional block skip switch on the machine operator's panel is set to on.
(See Section 13.2 for optional block skip.)

12.3 SUBPROGRAM

If a program contains a fixed sequence or frequently repeated pattern, such a sequence or pattern can be stored as a subprogram in memory to simplify the program.
A subprogram can be called from the main program.
A called subprogram can also call another subprogram.

Format

- **Subprogram configuration**

One subprogram

O □□□□ ; Subprogram number (or the colon (:) optionally in the case of ISO)

M99 ; Program end

M99 need not constitute a separate block as indicated below.

Example) **X100.0 Y100.0 M99 ;**

- **Subprogram call**

M98 P○○○ ○○○○ ;

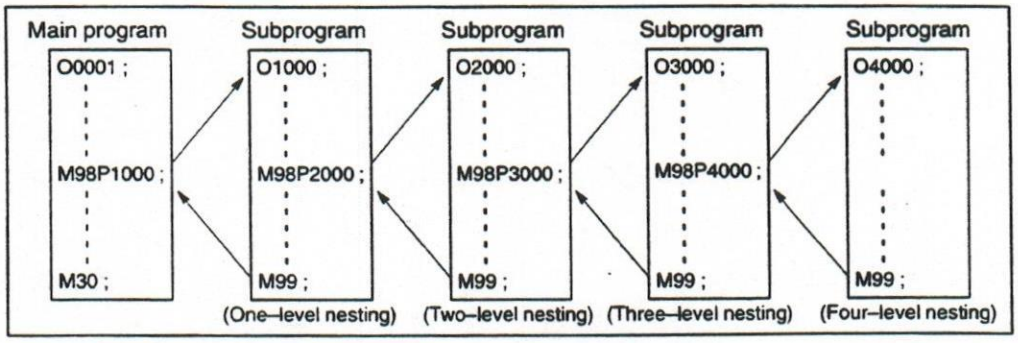
↑ Number of times the subprogram is called repeatedly

↑ Subprogram number

When no repetition data is specified, the subprogram is called just once.

Explanations

When the main program calls a subprogram, it is regarded as a one–level subprogram call. Thus, subprogram calls can be nested up to four levels as shown below.



A single call command can repeatedly call a subprogram up to 9999 times.
For compatibility with automatic programming systems, in the first block, Nxxxx can be used instead of a subprogram number that follows O (or :). A sequence number after N is registered as a subprogram number.

- **Reference** See Chapter 10 in Part III for the method of registering a subprogram.

Notes

Notes
1. The M98 and M99 code or strobe signals are not output to the machine tool.
2. If the subprogram number specified by address P cannot be found, an alarm (No. 078) is output.

Examples

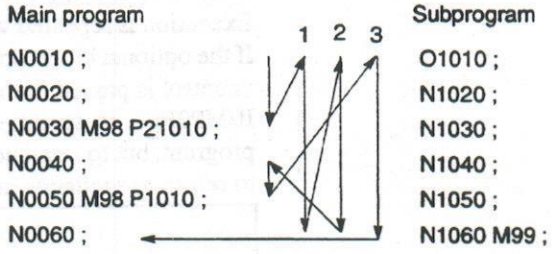
☆ **M98 P51002 ;**

This command specifies "Call the subprogram (number 1002) five times in succession." A subprogram call command (M98P_) can be specified in the same block as a move command.

☆ **X1000.0 M98 P1200 ;**

This example calls the subprogram (number 1200) after an X movement.

☆ Execution sequence of subprograms called from a main program

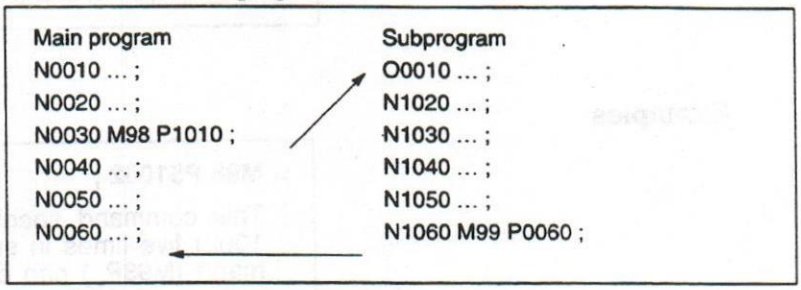


A subprogram can call another subprogram in the same way as a main program calls a subprogram.

Special Usage

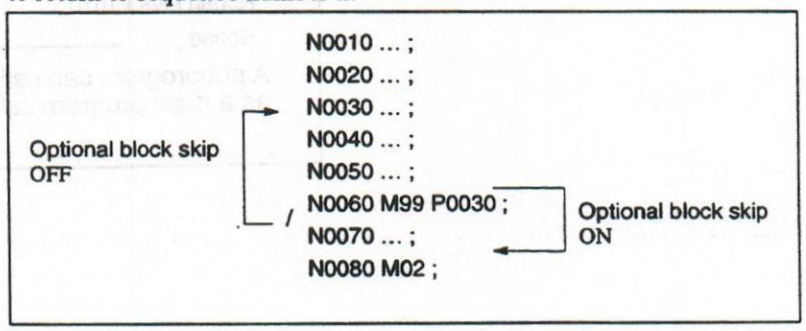
- **Specifying the sequence number for the return destination in the main program**

If P is used to specify a sequence number when a subprogram is terminated, control does not return to the block after the calling block, but returns to the block with the sequence number specified by P. Note, however, that P is ignored if the main program is operating in a mode other than memory operation mode.
This method consumes a much longer time than the normal return method to return to the main program.



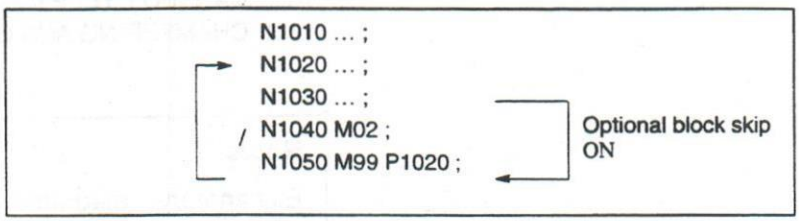
- **Using M99 in the main program**

If M99 is executed in a main program, control returns to the start of the main program. For example, M99 can be executed by placing /M99 ; at an appropriate location of the main program and setting the optional block skip function to off when executing the main program. When M99 is executed, control returns to the start of the main program, then execution is repeated starting at the head of the main program.
Execution is repeated while the optional block skip function is set to off. If the optional block skip function is set to on, the /M99 ; block is skipped ; control is passed to the next block for continued execution.
If/M99Pn ; is specified, control returns not to the start of the main program, but to sequence number n. In this case, a longer time is required to return to sequence number n.



- **Using a subprogram only**

A subprogram can be executed just like a main program by searching for the start of the subprogram with the MDI.
(See Section 9.4 in Part III for information about search operation.)
In this case, if a block containing M99 is executed, control returns to the start of the subprogram for repeated execution. If a block containing M99Pn is executed, control returns to the block with sequence number n in the subprogram for repeated execution. To terminate this program, a block containing /M02 ; or /M30 ; must be placed at an appropriate location, and the optional block switch must be set to off ; this switch is to be set to on first.



13 FUNCTIONS TO SIMPLIFY PROGRAMMING

General

This chapter explains the following items:

13.1 CANNED CYCLE (G90, G92, G94)
13.2 CHAMFERING AND CORNER R

Notes

Explanatory diagrams in this chapter uses diameter programming in X axis.

In radius programming, changes U/2 with U and X/2 with X.

13.1 CANNED CYCLE (G90, G92, G94)

There are three canned cycles : the outer diameter/internal diameter cutting canned cycle (G90), the thread cutting canned cycle (G92), and the end face turning canned cycle (G94).

13.1.1 Outer Diameter/Internal Diameter Cutting Cycle (G90)

- **Straight cutting cycle**

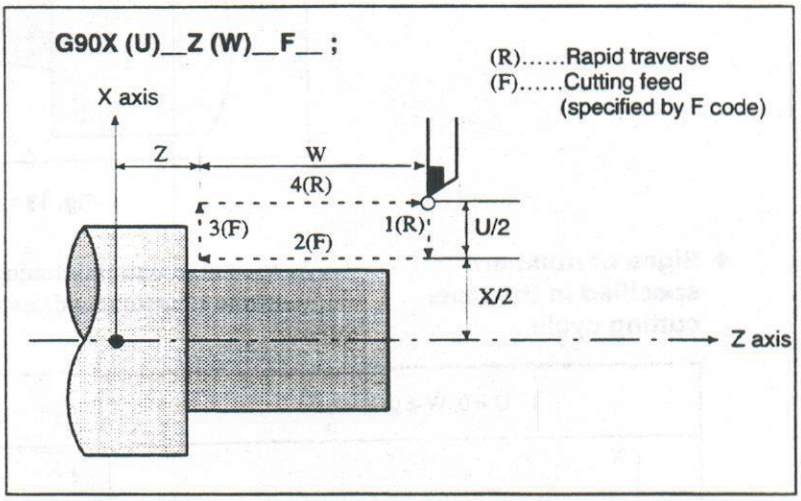


Fig.13.1.1 (a) Straight Cutting Cycle

In incremental programming, the sign of the numbers following address U and W depends on the direction of paths 1 and 2. In the cycle of 13. 1 1 (a), the signs of U and W are negative.
In single block mode, operations 1, 2, 3 and 4 are performed by pressing the cycle start button once.

- **Taper cutting cycle**

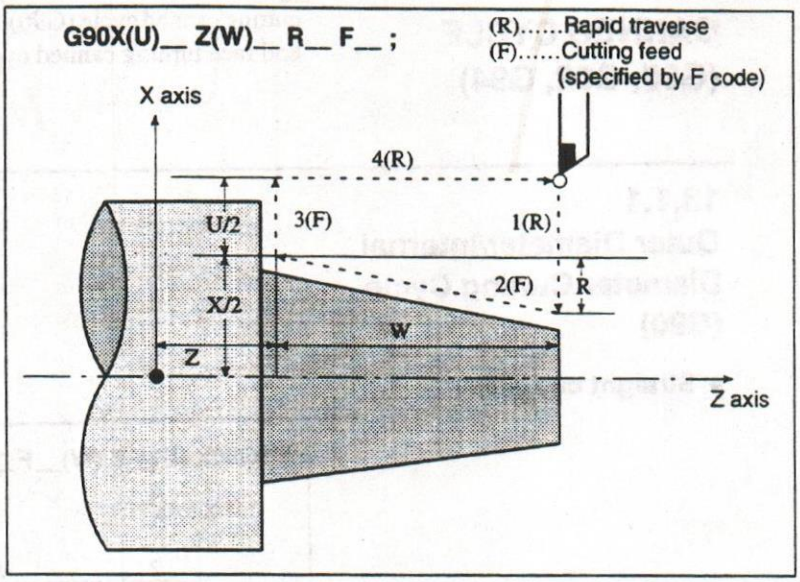
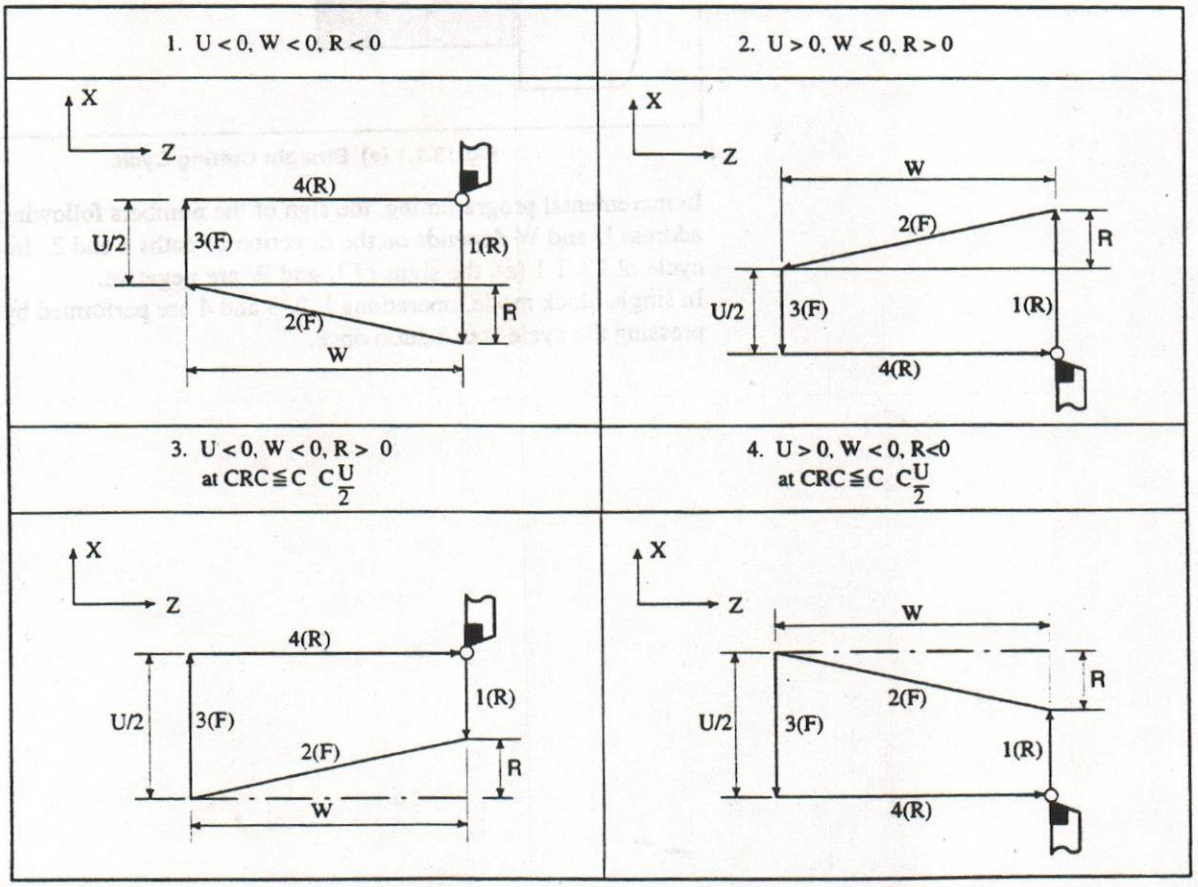


Fig. 13.1.1 (b) Taper Cutting Cycle

- **Signs of numbers specified in the taper cutting cycle**

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:



13.1.2 Thread Cutting Cycle (G92)

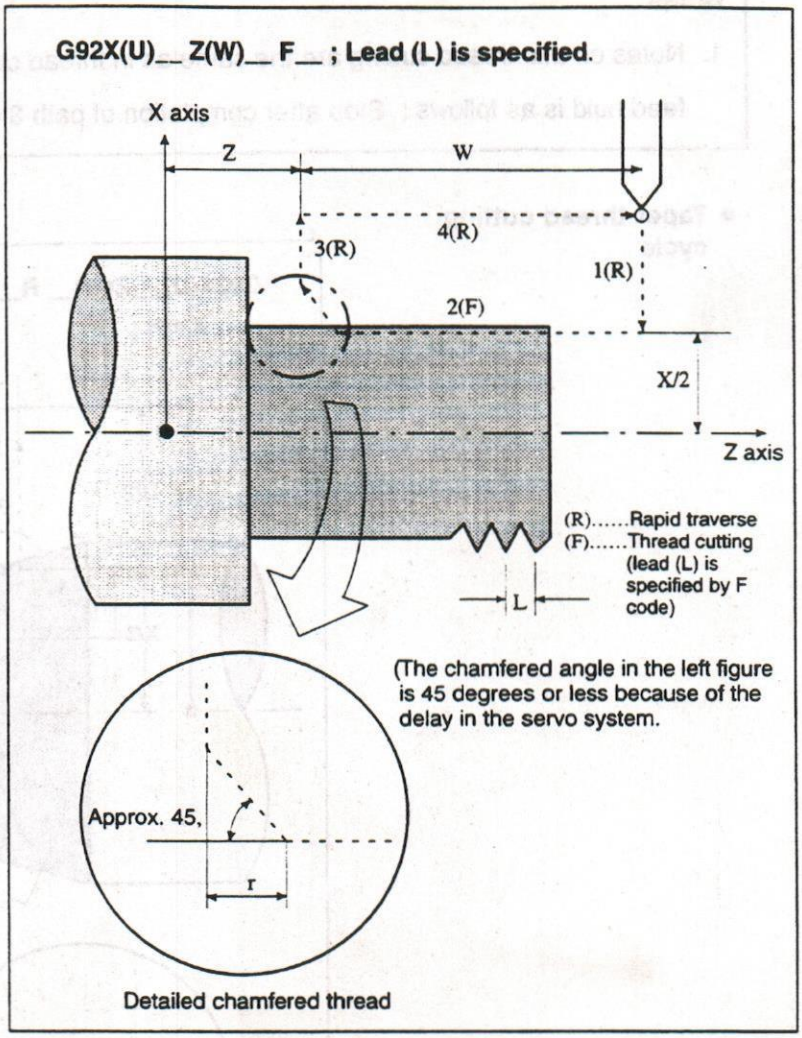


Fig. 13.1.2 (a) Straight Thread Cutting

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is , if the direction of path 1 is the negative along the X axis, the value of U is negative.
The range of thread leads, limitation of spindle speed, etc. are the same as in G32 (thread cutting). Thread chamfering can be performed in this thread cutting cycle. A signal from the machine tool, initiates thread chamfering. The chamfering distance is specified in a range from 0.1L to 12.7L in 0.1L increments by parameter (No. 5130). (In the above expression, L is the thread lead.)
In the single block mode, operations 1, 2, 3, and 4 are performed by pressing cycle start button once.

Notes

1. Notes on this thread cutting are the same as in thread cutting in G32. However, a stop by feed hold is as follows ; Stop after completion of path 3 of thread cutting cycle.

- **Taper thread cutting cycle**

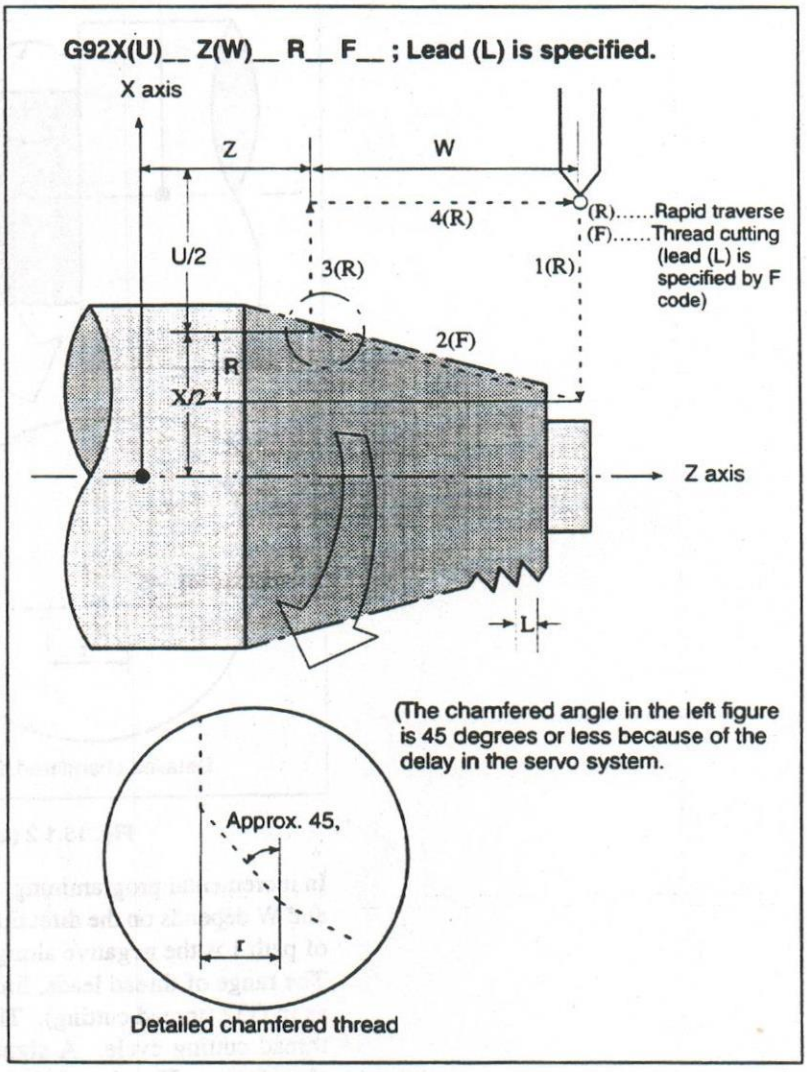


Fig. 13.1.2 (b) Taper thread cutting cycle

13.1.3 End Face Turning Cycle (G94)

- **Face cutting cycle**

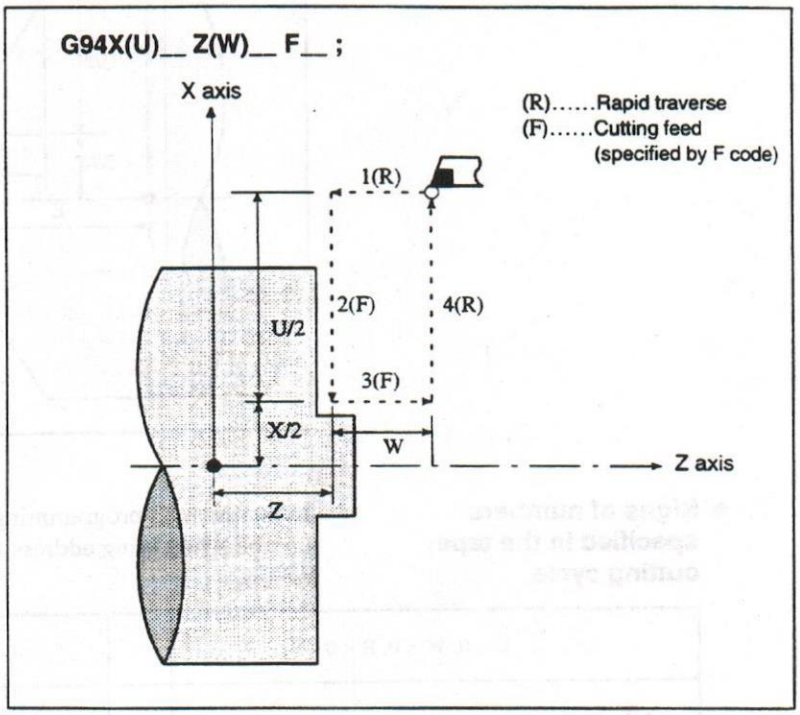


Fig. 13.1.3 (a) Face Cutting Cycle

In incremental programming, the sign of numbers following addresses U and W depends on the direction of paths 1 and 2. That is, if the direction of the path is in the negative direction of the Z axis, the value of W is negative.

In single block mode, operations 1, 2, 3, and 4 are performed by pressing the cycle start button once.

- **Taper face cutting cycle**

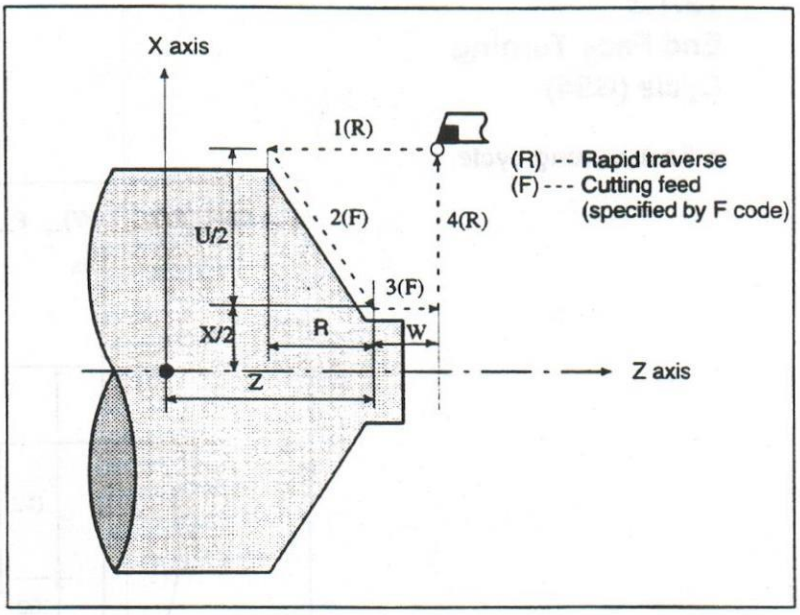
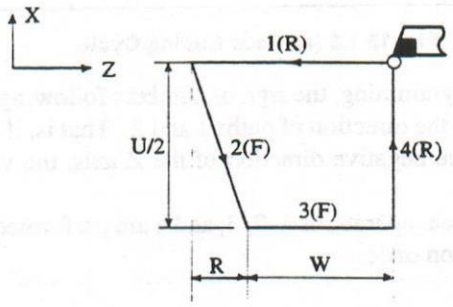
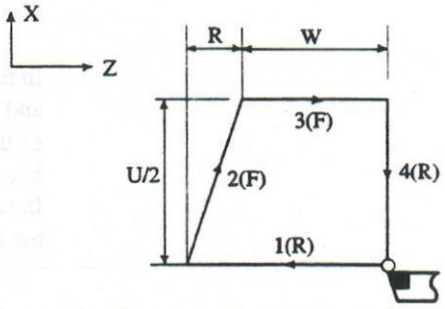
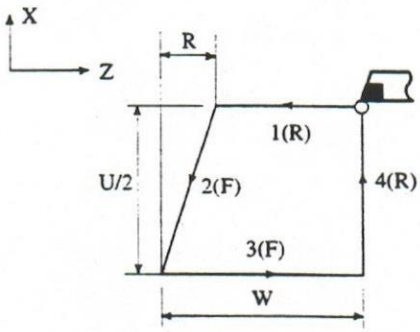
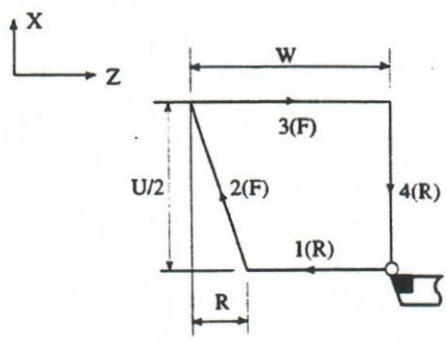


Fig. 13.1.3 (b)

- **Signs of numbers specified in the taper cutting cycle**

In incremental programming, the relationship between the signs of the numbers following address U, W, and R, and the tool paths are as follows:

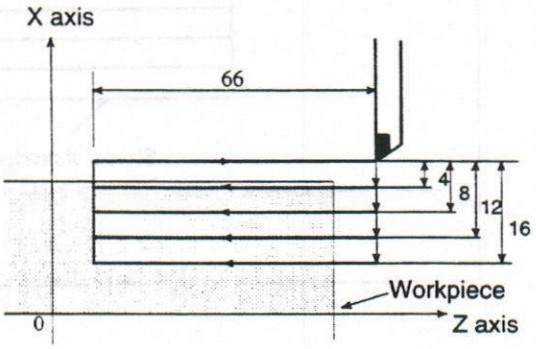
1. U < 0, W < 0, R < 0	2. U > 0, W < 0, R < 0
	
3. U < 0, W < 0, R > 0 at CRC≦CWC	**4. U > 0, W < 0, R<0 at CRC≦CWC**
	

Notes

1.Since data values of X (U), Z (W) and R during canned cycle are modal, if X (U), Z (W), or R is not newly commanded, the previously specified data is effective. Thus, when the Z axis movement amount does not vary as in the example below, a canned cycle can be repeated only by specifying the movement commands for the X–axis.

However, these data are cleared, if a one–shot G code expect for G04 (dwell) or a G code in the group 01 except for G90, G92, G94 is commanded.

(Example)



The cycle in the above figure is executed by the following program.

```
N030  G90 U–8.0 W–66.0 F0.4 ;
N031  U–16.0 ;
N032  U–24.0 ;
N033  U–32.0 ;
```

1.The following two applications can be performed.

(1) If an EOB or zero movement commands are specified for the block following that specified with a canned cycle, the same canned cycle is repeated.

(2) If the M, S, T function is commanded during the canned cycle mode, both the canned cycle and M, S, or T function can be performed simultaneously. If this is inconvenient, cancel the canned cycle once as in the program examples below (specify G00 or G01) and execute the M, S, or T command. After the execution of M, S, or T terminates, command the canned cycle again.

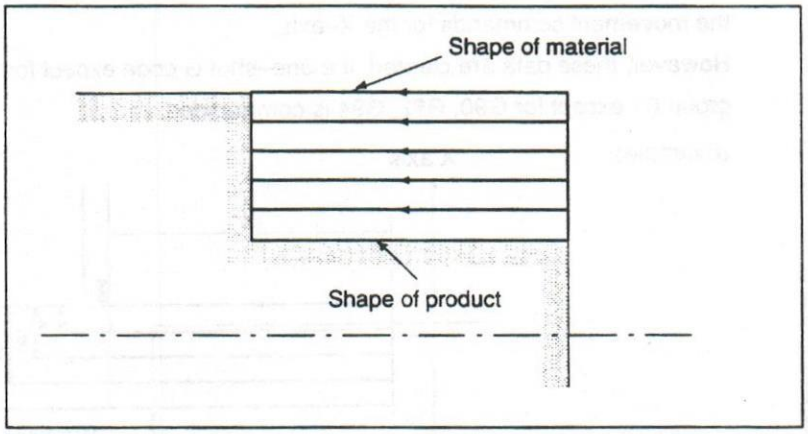
(Example)

```
N003 T0101 ;
  :
  :
N010  G90 X20.0 Z10.0 F0.2 ;
N011  G00 T0202 ;
N012  G90 X20.5 Z10.0 ;
```

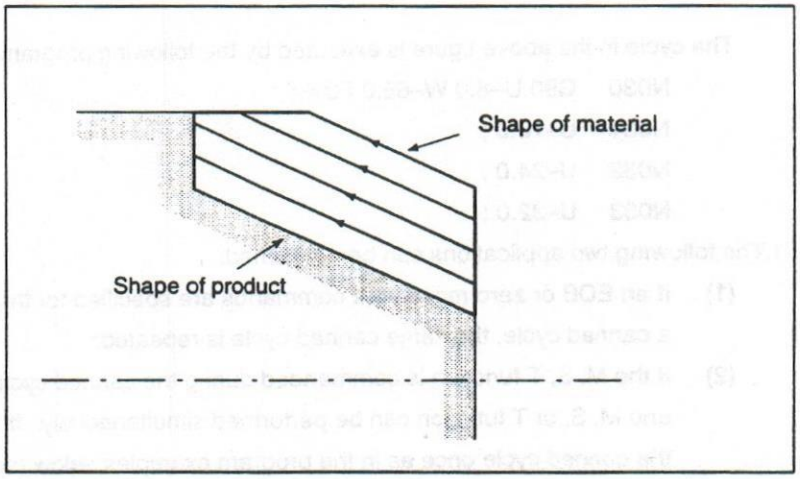
13.1.4 How to Use Canned Cycles (G90, G92, G94)

An appropriate canned cycle is selected according to the shape of the material and the shape of the product.

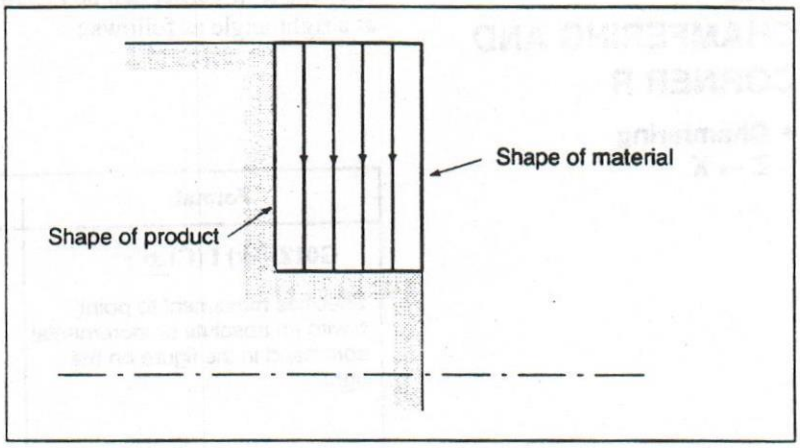
- **Straight cutting cycle (G90)**



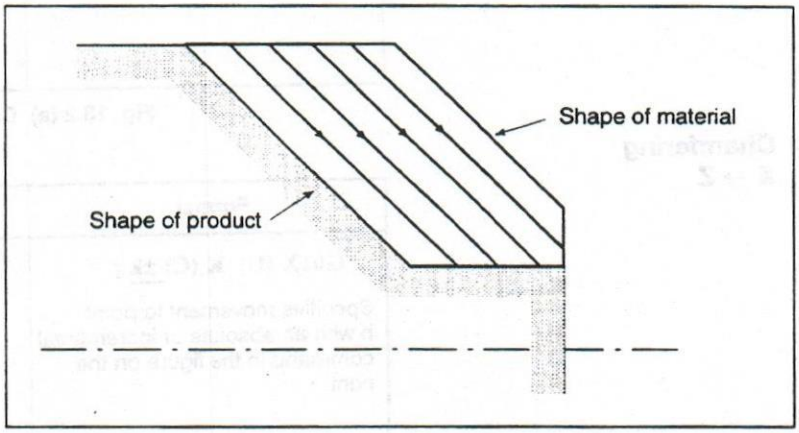
- **Taper cutting cycle (G90)**



- **Face cutting cycle (G94)**



- **Face taper cutting cycle (G94)**



13.2 CHAMFERING AND CORNER R

A chamfer or corner can be inserted between two blocks which intersect at a right angle as follows :

- **Chamfering**
  Z → X

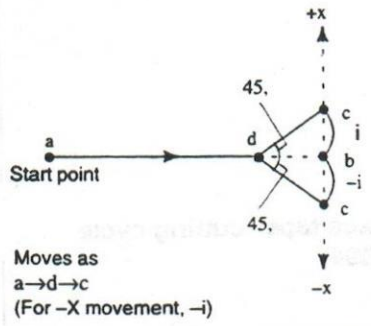
Format	Tool movement
G01Z (W) I (C) ±i ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

Fig. 13.2 (a) Chamfering (Z→X)

- **Chamfering**
  X → Z

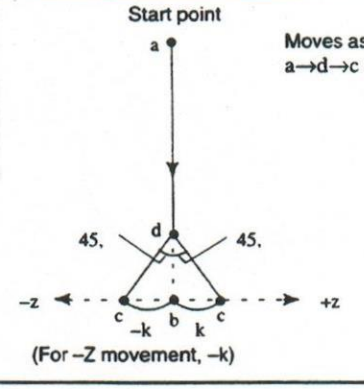
Format	Tool movement
G01X (U) K (C) ±k ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

Fig. 13.2 (b) Chamfering (X→Z)

- **Corner R**
  Z → X

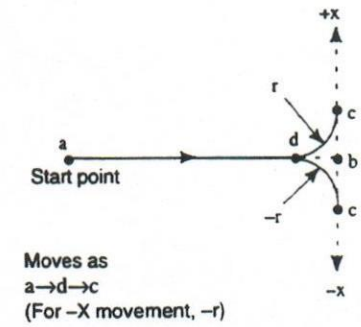
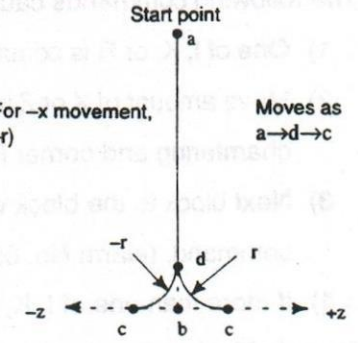
Format	Tool movement
G01Z (W) R ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

Fig. 13.2 (c) Corner R (Z→X)

- **Corner R**
  X → Z

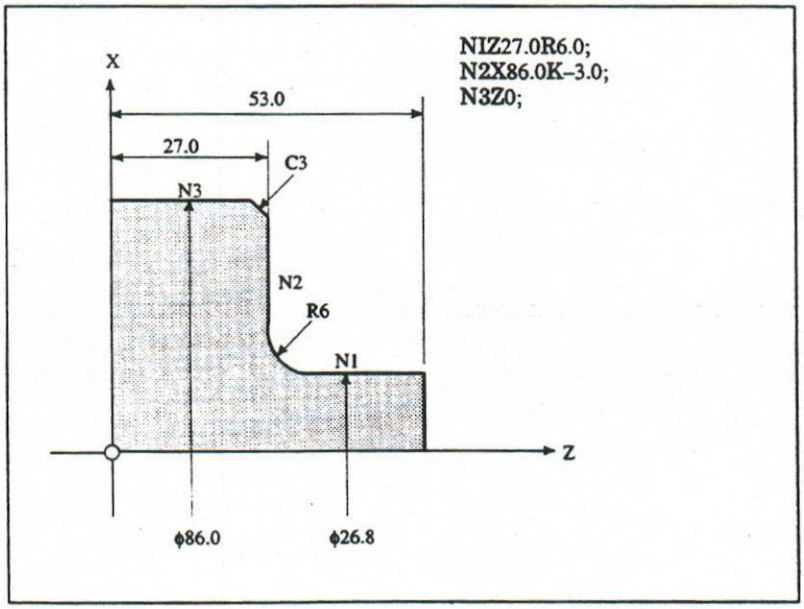
Format	Tool movement
G01X (U) R ±r ; Specifies movement to point b with an absolute or incremental command in the figure on the right.	

13.2 (d) Corner R (X→Z)

Explanations

The movement for chamfering or corner R must be a single movement along the X or Z axis in G01 mode. The next block must be a single movement along the X or Z axis perpendicular to the former block.
I or K, and R always specify a radius value.
Note that the start point for a command specified in a block following a chamfering or corner–R block is not point c but point b shown in Figs. 13.2 (a) to (d). In incremental programming, specify a distance from point b.

Examples



Notes

1.The following commands cause an alarm.

1) One of I, K, or R is commanded when X and Z axes are specified by G01. (alarm No. 054)
2) Move amount of X or Z is less than chamfering value and corner R value in the block where chamfering and corner R are specified. (alarm No. 055)
3) Next block to the block where chamfering and corner R were specified, has not G01 command. (alarm No. 051, 052)
4) If more than one of I, K, and R re specified in G01, alarm No. 053 is issued.

2.A single block stops at point c of Fig. 15.5 (a) A 15.5 (d), not at point d.

3.Chamfering and corner R cannot be applied to a thread cutting block.

4.C can be used instead of I or K as an address for chamfering. To use C for an address for chamfering, fix parameter CCR No. 3405#4 to 1.

5.If both C and R are specified with G01 in a block, the address specified last is valid.

14 COMPENSATION FUNCTION

This chapter describes the following compensation functions:

14.1 TOOL OFFSET
14.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION
14.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION
14.4 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

14.1 TOOL OFFSET

Tool offset is used to compensate for the difference when the tool actually used differs from the imagined tool used in programming (usually, standard tool).

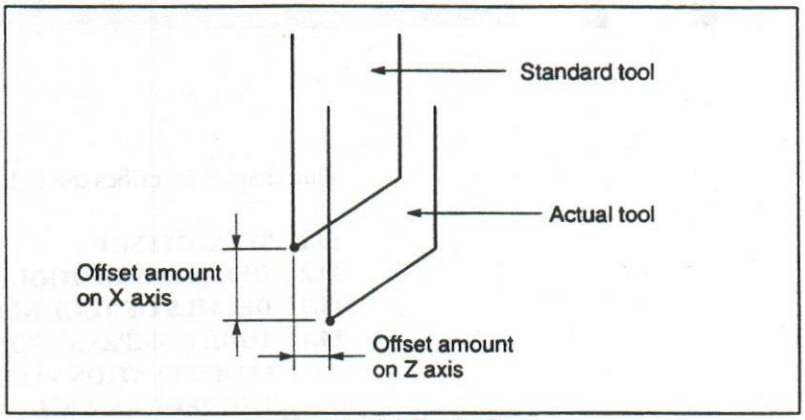


Fig.14.1(a) Tool offset

In this unit, there is no G code to specify tool offset.
The tool offset is specified by T code.

14.1.1 Tool Geometry Offset and Tool Wear Offset

Tool geometry offset and tool wear offset are possible to divide the tool offset to the tool geometry offset for compensating the tool shape or tool mounting position and the tool wear offset for compensating the tool nose wear.
Total value of tool geometry offset value and tool wear offset value is set as the tool wear offset value without option.

Notes

Tool geometry offset and tool wear offset are optioned.

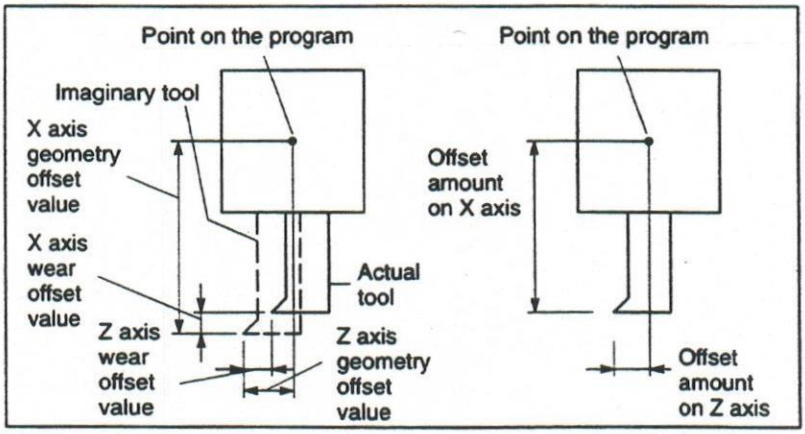

Point on the program
Imaginary tool
X axis geometry offset value
X axis wear offset value
Z axis wear offset value
Actual tool
Z axis geometry offset value
Point on the program
Offset amount on X axis
Offset amount on Z axis

Fig. 14.1.1(a) Difference the tool geometry offset from tool wear offset

Fig. 14.1.1(b) Not difference the tool geometry offset from tool wear offset

14.1.2 T Code for Tool Offset

There are two methods for specifying a T code as shown in Table 14.1.2(a) and Table 14.1.2(b).

Format

- **Lower digit of T code specifies geometry and wear offset number**

Table 14.1.2(a)

Kind of T code	Meaning of T code	Parameter setting for specifying of offset No.	
2-digit command	T○○ Tool wear and tool geometry offset number Tool selection	When LD1, bit 0 of parameter No.5002, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When LGN, bit 1 of parameter No.5002, is set to 0, the tool geometry offset number and tool wear offset number specified for a certain tool are the same.
4-digit command	T○○○○ Tool wear and tool geometry offset number Tool selection	When LD1, bit 0 of parameter No.5002, is set to 0, a tool wear offset number is specified with the last two digits of a T code.	

- **Lower digit of T code specifies wear offset number and higher digit number specifies tool selection number and geometry offset number**

Table 14.1.2(b)

Kind of T code	Meaning of T code	Parameter setting for specifying of offset No.	
2-digit command	T○○ Tool wear offset number Tool selection and tool geometry offset number	When LD1, bit 0 of parameter No.5002, is set to 1, a tool wear offset number is specified with the last digit of a T code.	When LGN, bit 1 of parameter No.5002, is set to 0, the tool geometry offset number and tool selection number specified for a certain tool are the same.
4-digit command	T○○○○ Tool wear offset number Tool selection and tool geometry offset number	When LD1, bit 0 of parameter No.5002, is set to 0, a tool wear offset number is specified with the last two digits of a T code.	

14.1.3 Tool Selection

Tool selection is made by specifying the T code corresponding to the tool number. Refer to the machine tool builder's manual for the relationship between the tool selection number and the tool.

14.1.4 Offset Number

Tool offset number has two meanings.
It is specifies the offset distance corresponding to the number that is selected to begin the offset function. A tool offset number of 0 indicates that the offset amount is 0 and the offset is cancelled.

14.1.5 Offset

There are two types of offset. One is tool wear offset and the other is tool geometry offset.

Explanations

- **Tool wear offset**

The tool path is offset by the X and Z wear offset values for the programmed path. The offset distance corresponding to the number specified by the T code is added to or subtracted from the and position of each programmed block.

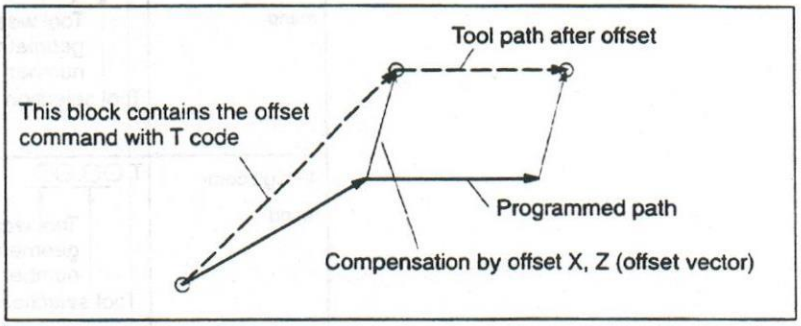

Tool path after offset
This block contains the offset command with T code
Programmed path
Compensation by offset X, Z (offset vector)

Fig.14.1.5(a) Movement of offset (1)

- **Offset vector**

In Fig.14.1.5(a), the vector with offset X and Z is called the offset vector. Compensation is the same as the offset vector.

- **Tool wear offset cancel**

Offset is cancelled when T code offset number 0 is selected. At the end of the cancelled block, the offset vector becomes 0.

N1 X60.0 Z50.0 T0202 ; Creates the offset vector corresponding to offset number 02

N2 Z200.0 ;

N3 X200.0 Z150.0 T0200 ; Specifying offset number 0 deletes the offset vector.

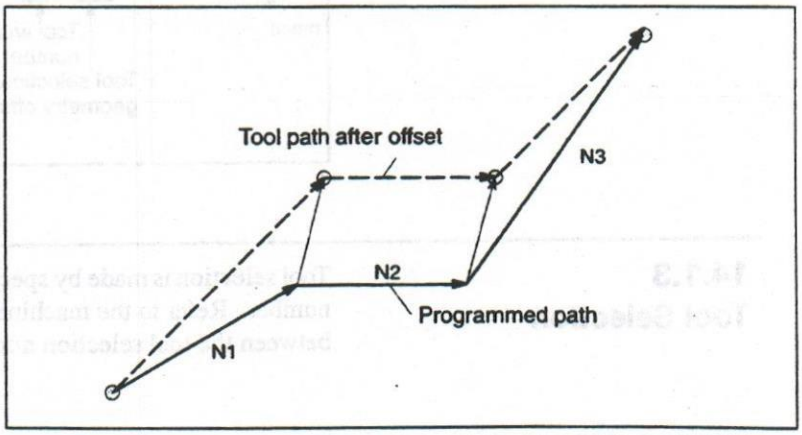


Fig.14.1.5(b) Movement of offset (2)

Bit 6 (LVC) of parameter No.5003 can specify that the offset be canceled at a reset.

- **Tool wear offset when only a T code is specified**

When only a T code is specified in a block, the tool is moved by the wear offset value without a move command. The movement is performed at rapid traverse rate in the G00 mode . It is performed at feedrate in other modes.
When a T code with offset number 0 is specified by itself, movement is performed to cancel the offset.

Note
When G50 X_Z_T_ ; is specified
Tool is not moved.
The coordinate system in which the coordinate value of the tool position is (X,Z) is set. The tool position is obtained by subtracting the wear offset value corresponding to the offset number specified in the T code.

- **Tool geometry offset**

With the tool geometry offset, the work coordinate system is shifted by the X and Z geometry offset amounts. Namely, the offset amount corresponding to the number designated with the code is added to or subtracted from the current position.

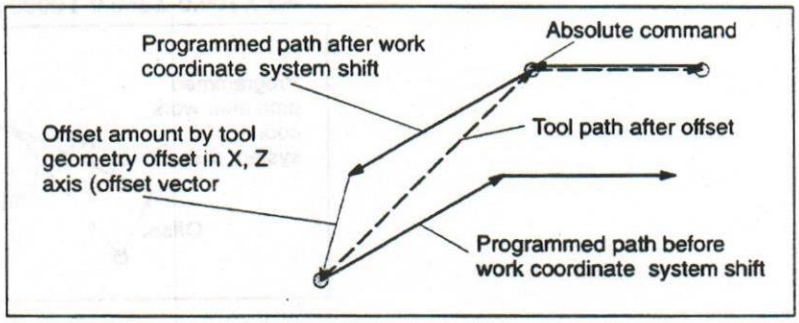

Programmed path after work coordinate system shift
Absolute command
Offset amount by tool geometry offset in X, Z axis (offset vector
Tool path after offset
Programmed path before work coordinate system shift

Fig.14.1.5(c) Movement of tool geometry offset

Note
As well as wear offset, the tool can be compensated by parameter setting LGT(No.5002#4) to add or subtract the programmed end point of each block.

- **Tool geometry offset cancel**

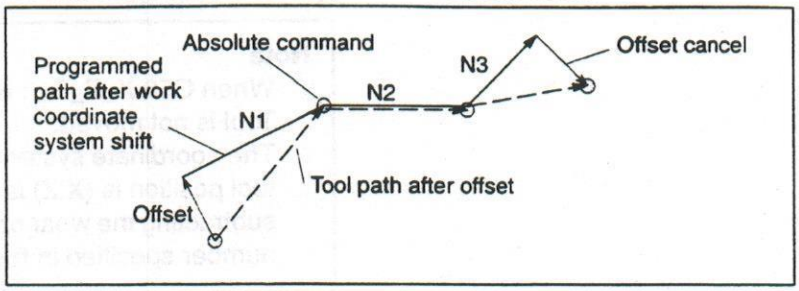
Specifying offset number 0 cancels offset.
Bit 7 (TGC) of parameter No.5003 can specify that the tool geometry offset be canceled at a reset.

Note
If the same number is used to specify the tool geometry offset and tool wear offset (bit 1 (LGN) of parameter No.5002 is 0), specifying a T code with offset No. 0 does not cancel the tool geometry offset.
However, bit 5 (LGC) of parameter No.5002 can specify that the tool geometry offset be canceled by offset No.0.

Examples

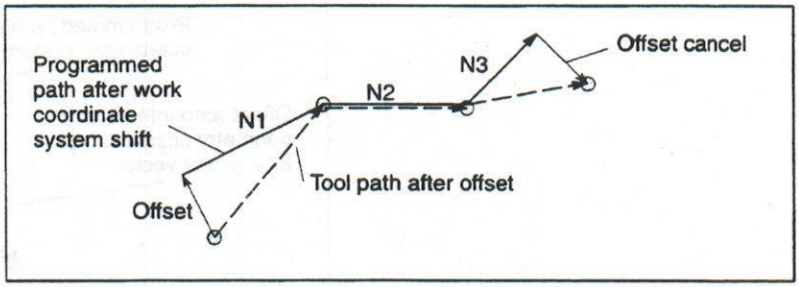
1. When a tool geometry offset number and tool wear offset number are specified with the last two digits of a T code (when LGN, bit 1 of parameter No.5002, is set 0),

N1 X50.0 Z100.0 T0202 ; Specifies offset number 02
N2 Z200.0 ;
N3 X100.0 Z250.0 T0200 ; Cancels offset



2. Assume that geometry offset is not cancelled with offset No.0 (Set the parameter (No.5002#1).)

N1 X50.0 Z100.0 T0202 ; Tool selection number (specified tool geometry offset number 02)
N2 Z200.0 ;
N3 X100.0 Z250.0 T0000 ; Cancels offset



14.2 OVERVIEW OF TOOL NOSE RADIUS COMPENSATION

It is difficult to produce the compensation necessary to form accurate parts when using only the tool offset function due to tool nose roundness in taper cutting or circular cutting. The tool nose radius compensation function compensates automatically for the above errors.

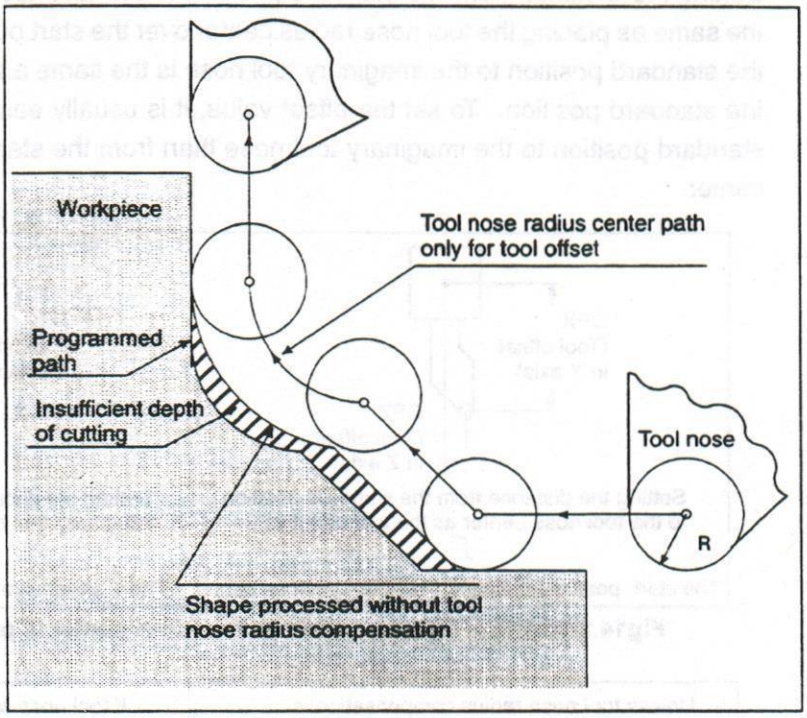


Fig14.2 Tool path of tool nose radius compensation

14.2.1 Imaginary Tool Nose

The tool nose at position A in following figure does not actually exist. The imaginary tool nose is required because it is usually more difficult to set the actual tool nose radius center to the start position than the imaginary tool nose.

Also when imaginary tool nose is used, the tool nose radius need not be considered in programming.

The position relationship when the tool is set to the start position is shown in the following figure.

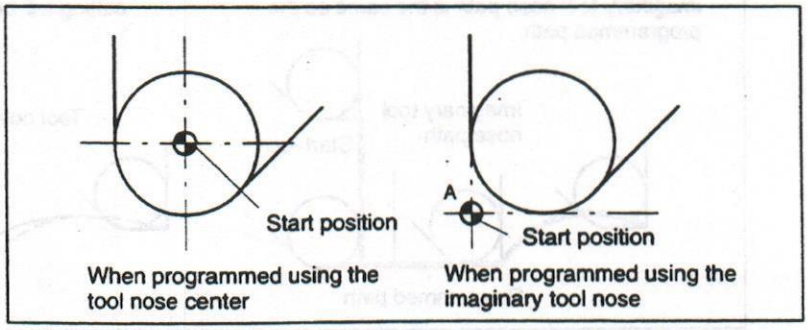


Fig.14.2.1(a) Tool nose radius center and imaginary tool nose

Notes

In a machine with reference positions, a standard position like the turret center can be placed over the start position. The distance from this standard position to the nose radius center or the imaginary tool nose is set as the tool offset value.

Setting the distance from the standard position to the tool nose radius center as the offset value is the same as placing the tool nose radius center over the start position, while setting the distance from the standard position to the imaginary tool nose is the same as placing the imaginary tool nose over the standard position. To set the offset value, it is usually easier to measure the distance from the standard position to the imaginary tool nose than from the standard position to the tool nose radius center.

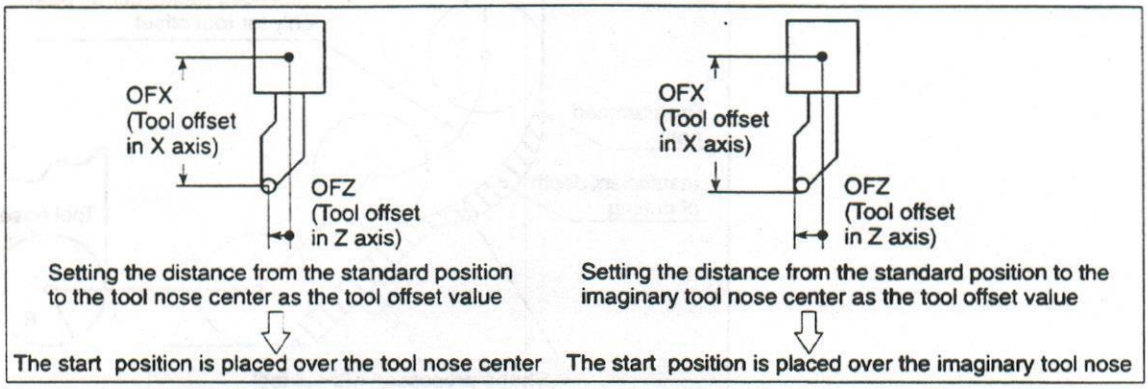

OFX
(Tool offset
in X axis)
OFZ
(Tool offset
in Z axis)
Setting the distance from the standard position to the tool nose center as the tool offset value
The start position is placed over the tool nose center
OFX
(Tool offset
in X axis)
OFZ
(Tool offset
in Z axis)
Setting the distance from the standard position to the imaginary tool nose center as the tool offset value
The start position is placed over the imaginary tool nose

Fig14.2.1(b) Tool offset value when the turret center is placed over the start position

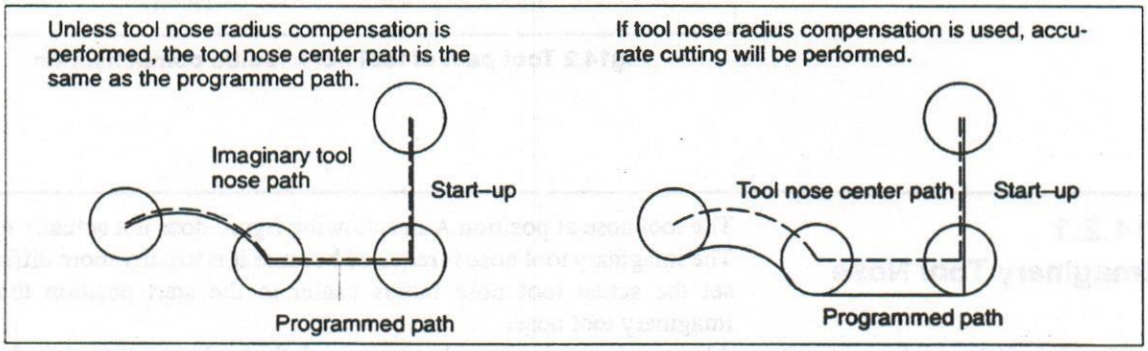

Unless tool nose radius compensation is performed, the tool nose center path is the same as the programmed path.
Imaginary tool nose path
Start–up
Programmed path
If tool nose radius compensation is used, accurate cutting will be performed.
Tool nose center path
Start–up
Programmed path

Fig14.2.1(c) Tool path when programming using the tool nose center

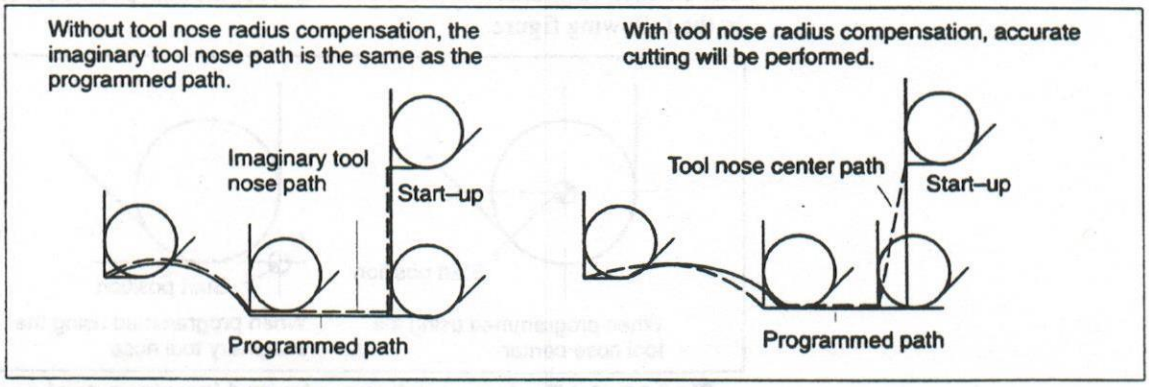

Without tool nose radius compensation, the imaginary tool nose path is the same as the programmed path.
Imaginary tool nose path
Start–up
Programmed path
With tool nose radius compensation, accurate cutting will be performed.
Tool nose center path
Start–up
Programmed path

Fig14.2.1(d) Tool path when programming using the imaginary tool nose

14.2.2 Direction of Imaginary Tool Nose

The direction of the imaginary tool nose viewed from the tool nose center is determined by the direction of the tool during cutting, so it must be set in advance as well as offset values.

The direction of the imaginary tool nose can be selected from the eight specifications shown in the Fig.14.2.2 below together with their corresponding codes.

This Fig14.2.2 illustrates the relation between the tool and the start position. The following apply when the tool geometry offset and tool wear offset option are selected.

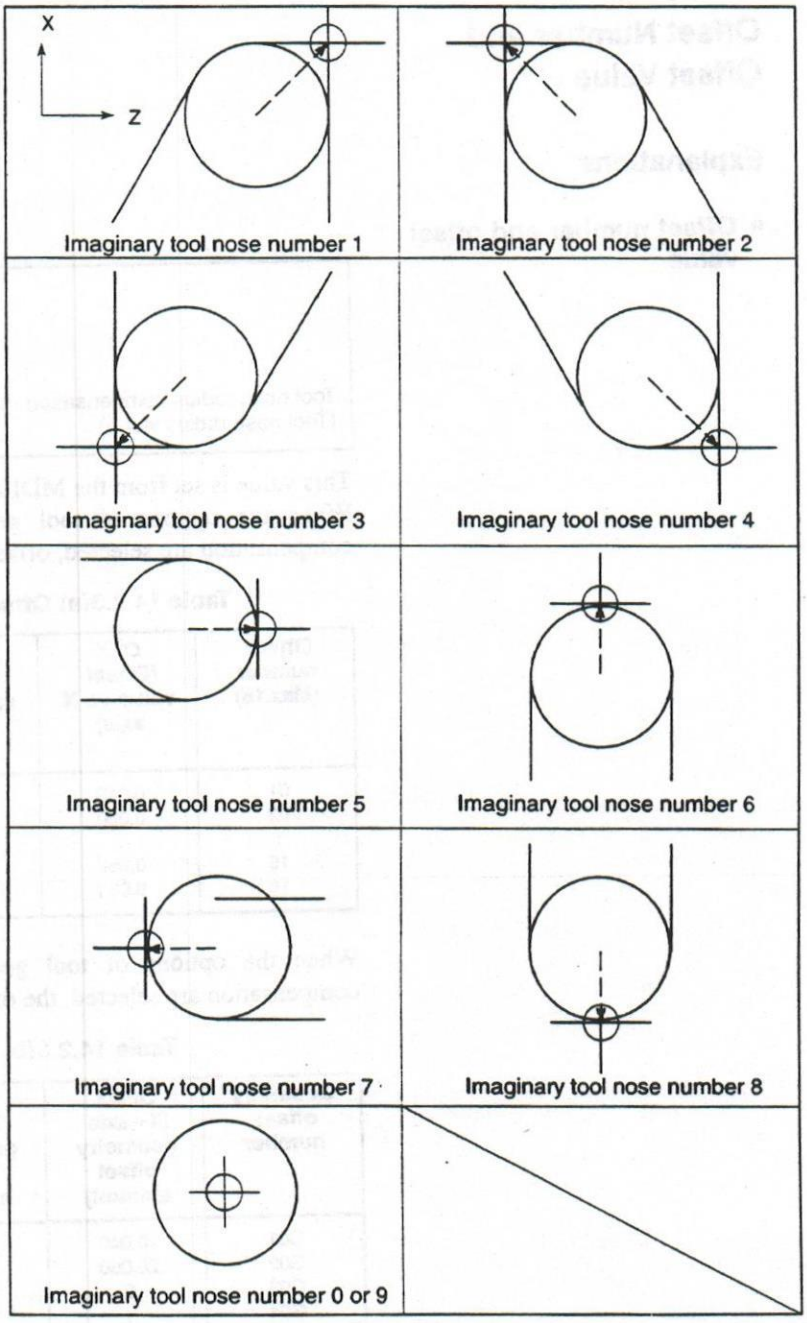

X
Z
Imaginary tool nose number 1
Imaginary tool nose number 2
Imaginary tool nose number 3
Imaginary tool nose number 4
Imaginary tool nose number 5
Imaginary tool nose number 6
Imaginary tool nose number 7
Imaginary tool nose number 8
Imaginary tool nose number 0 or 9

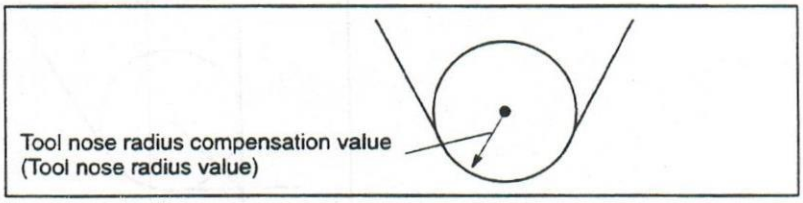
Fig.14.2.2 Direction of imaginary tool nose

Imaginary tool nose numbers 0 and 9 are used when the tool nose center coincides with the start position. Set imaginary tool nose number to address OFT for each offset number.
Bit 7 (WNP) of parameter No. 5002 is used to determine whether the tool geometry offset number or the tool wear offset number specifies the direction of the virtual tool nose for tool nose radius compensation.

14.2.3 Offset Number and Offset Value

Explanations

- **Offset number and offset value**



This value is set from the MDI according to the offset number.
When the options of tool geometry compensation and tool wear compensation are selected, offset values become as follows :

Table 14.2.3(a) Offset number and offset value

Offset number (Max.16)	OFX (Offset value on X axis)	OFZ (Offset value on Z axis)	OFR (Tool nose radius compensation value)	OFT (Direction of imaginary tool nose)
01	0.040	0.020	0.20	1
02	0.060	0.030	0.25	2
:	:	:	:	:
15	0.050	0.015	0.12	6
16	0.030	0.025	0.24	3

When the options of tool geometry compensation and tool wear compensation are selected, the offset values become as follows :

Table 14.2.3(b) Tool geometry offset

Geometry offset number	OFGX (X-axis geometry offset amount)	OFGZ (Z-axis geometry offset amount)	OFGR (Tool nose radius geometry offset value)	OFT (Imaginary tool nose direction)
G01	10.040	50.020	0	1
G02	20.060	30.030	0	2
G03	0	0	0.20	6
G04	:	:	:	:
G05	:	:	:	:
:	:	:	:	:

Table 14.2.3(c) Tool wear offset

Wear offset number	OFGX (X-axis wear offset amount)	OFGZ (Z-axis wear offset amount)	OFGR (Tool nose radius wear offset value)	OFT (Imaginary tool nose direction)
W01	0.040	0.020	0	1
W02	0.060	0.030	0	2
W03	0	0	0.20	6
W04	:	:	:	:
W05	:	:	:	:
:	:	:	:	:

- **Tool nose radius compensation**

In this case, the tool nose radius compensation value is the sum of the geometry or the wear offset value.

OFR=OFGR+OFWR

- **Imaginary tool nose direction**

The imaginary tool nose direction may be set for either the geometry offset or the wear offset.
However, the last designated direction later is effective.

- **Command of offset value**

A offset number is specified with the same T code as that used for tool offset. For details, see 14.1.2.

Notes

When the geometry offset number is made common to the tool selection by the parameter setting and a T code for which the geometry offset and wear offset number differ from each other is designated, the imaginary tool nose direction specified by the geometry offset number is valid.

Example) T0102
$OFR=RFGR_{01}+OFWR_{02}$
$OFT=OFT_{01}$

However bit 7 (WWP) of parameter No.5002 can specify that the imaginary tool nose direction specified with a wear offset number be effective.

- **Setting range of offset value**

The range of the offset value is an follows :

Increment system	metric system	Inch system
IS-B	0 to ±999.999 mm	0 to ±99.9999 inch

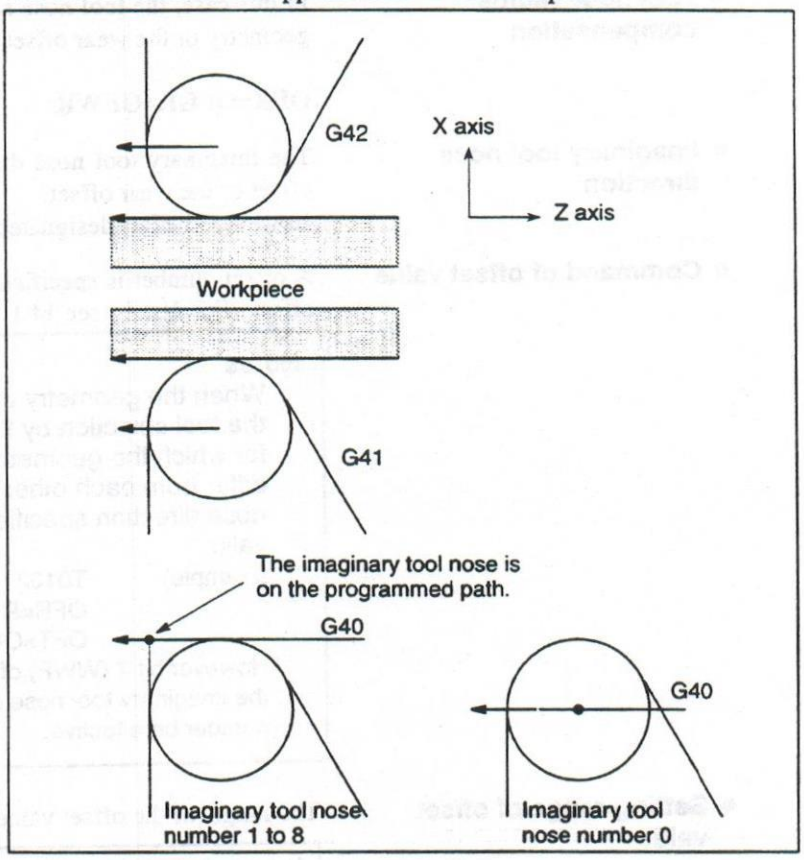
The offset value corresponding to the offset number 0 is always 0.
No offset value can be set to offset number 0.

14.2.4 Work Position and Move Command

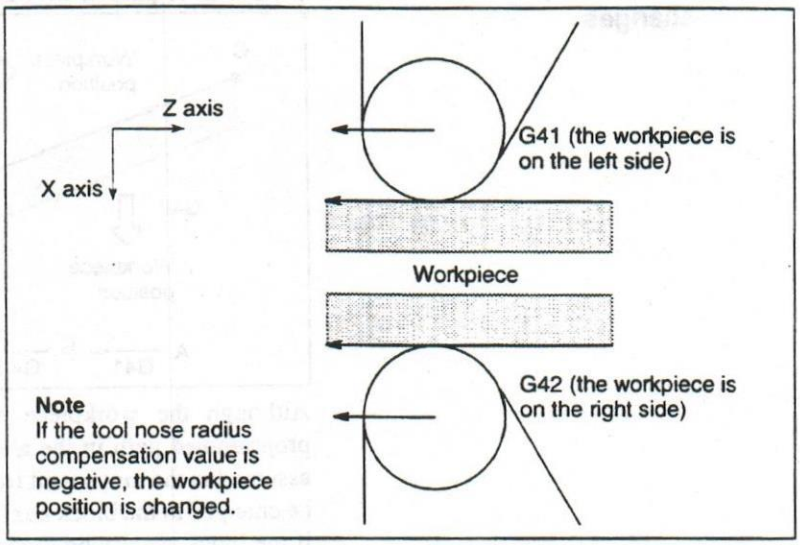
In tool nose radius compensation, the position of the workpiece with respect to the tool must be specified.

G code	Workpiece position	Tool path
G40	(Cancel)	Moving along the programmed path
G41	Right side	Moving on the left side the programmed path
G42	Left side	Moving on the right side the programmed path

The tool is offset to the opposite side of the workpiece.



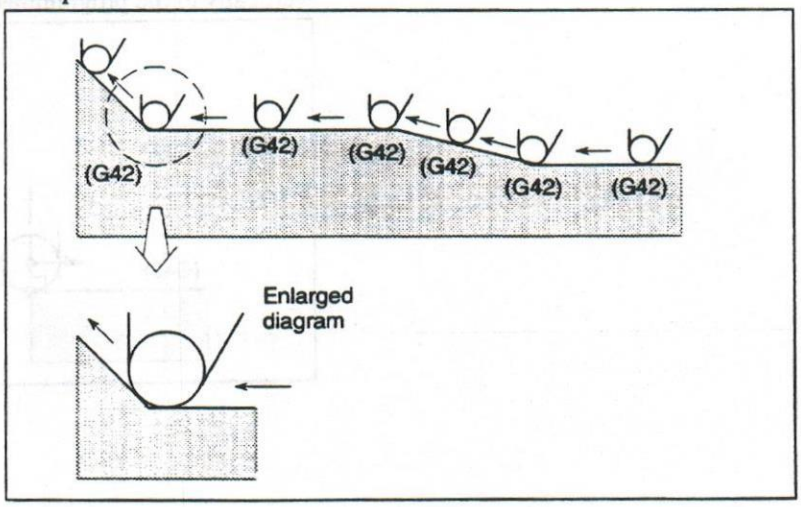
The workpiece position can be changed by setting the coordinate system as shown below.


Z axis
X axis
G41 (the workpiece is on the left side)
Workpiece
G42 (the workpiece is on the right side)
Note
If the tool nose radius compensation value is negative, the workpiece position is changed.

G40, G41, and, G42 are modal.
Don't specify G41 while in the G41 mode. If you do, compensation will not work properly.
Don't specify G42 while in the G42 mode for the same reason.
G41 or G42 mode blocks in which G41 or G42 are not specified are expressed by (G41) or (G42) respectively.

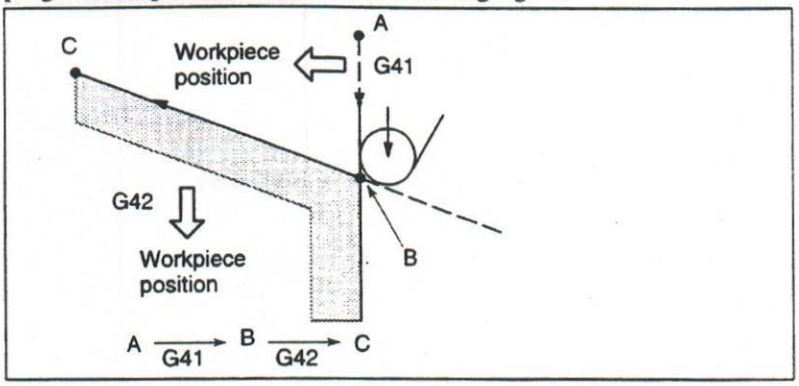
- **Tool movement when the workpiece position does not change**

When the tool is moving, the tool nose maintains contact with the workpiece.



- **Tool movement when the workpiece position changes**

The workpiece position against the toll changes at the corner of the programmed path as shown in the following figure.



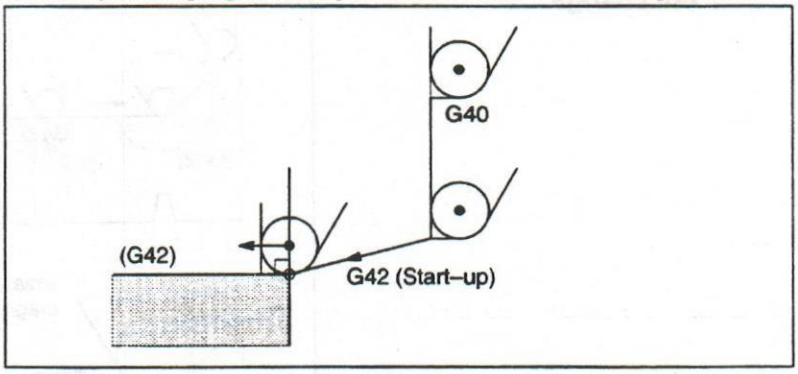
Although the workpiece does not exist on the right side of the programmed path in the above case. the existence of the workpiece is assumed in the movement from A to B. The workpiece position must not be changed in the block next to the start–up block. In the above example, if the block specifying motion from A to B were the start–up block, the tool path would not be the same as the one shown.

- **Start–up**

The block in which the mode changes to G41 or G42 from G40 is called the start–up block.

G40 _ ;
G41 _ ; (Start–up block)

Transient tool movements for offset are performed in the start–up block. In the block after the start–up block, the tool nose center is positioned Vertically to the programmed path of that block at the start position.

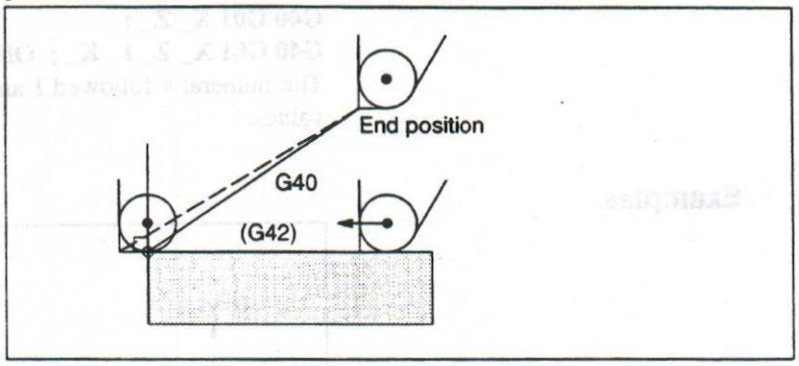


- **Offset cancel**

The block in which the mode changes to G40 from G41 or G42 is called the offset cancel block.

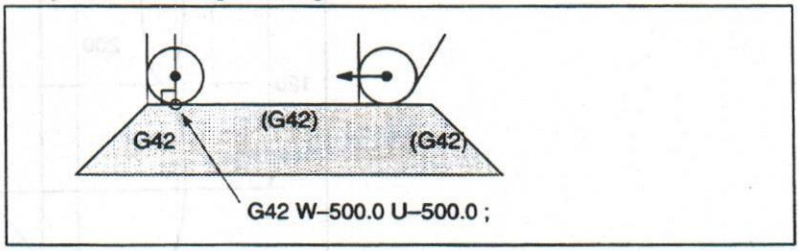
G41 _ ;
G40 _ ; (Offset cancel block)

The tool nose center moves to a position vertical to the programmed path in the block before the cancel block. The tool is positioned at the end position in the offset cancel block (G40) as shown below.



- **Specification of G41/G42 in G41/G42 mode**

When is specified again in G41/G42 mode , the tool nose center is positioned vertical to the programmed path of the preceding block at the end position of the preceding block.

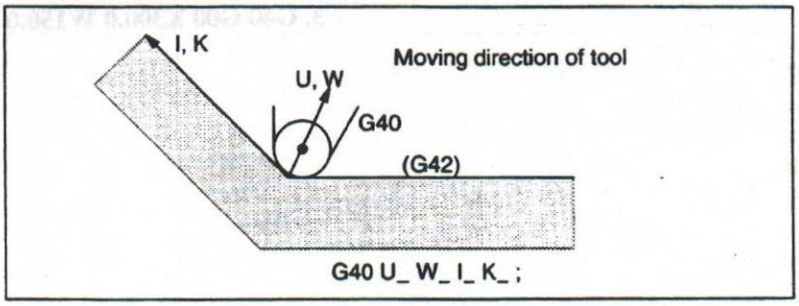


In the block that first specifies G41/G42, the above positioning of the tool nose center is not performed.

- **Tool movement when the moving direction of the tool in a block which includes a G40 command is different from the direction of the workpiece**

When you wish to retract the tool in the direction specified by X(U) and Z(W) cancelling the tool nose radius compensation at the end of machining the first block in the figure below, specify the following :

G40 X(U) _ Z(W) _ I _ K _ ;

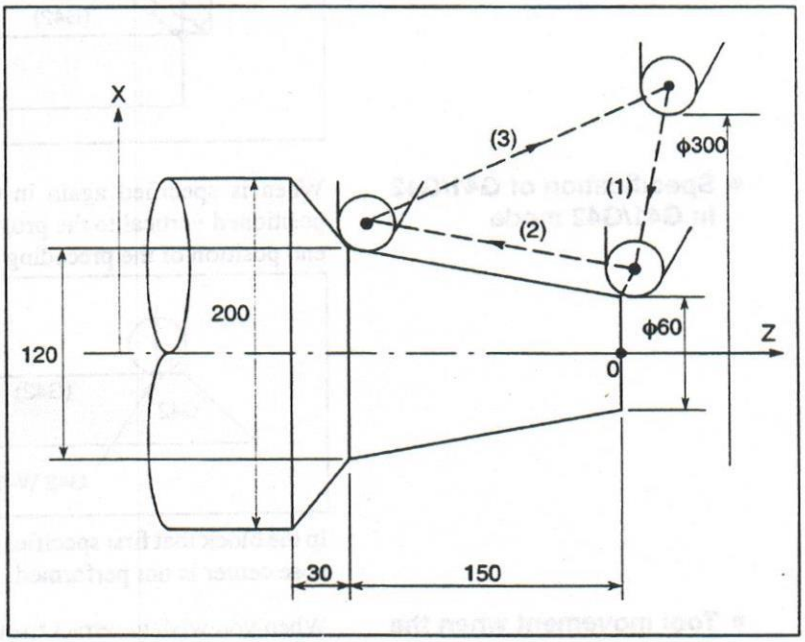


The workpiece position specified by addresses I and K is the same as that in the preceding block. If I and/or K is specified with G40 in the offset cancel mode, the I and/or K is ignored.

G40 X_ Z_ I_ K_ ;	Tool nose radius compensation
G40 G02 X_ Z_ I_ K_ ;	Circular interpolation

G40 G01 X_ Z_ ;
G40 G01 X_ Z_ I_ K_ ; Offset cancel mode (I and k are ineffective.)
The numeral s followed I and K should always be specified as radius values.

Examples



(G40 mode)
1. G42 G00 X60.0 ;
2. G01 X120.0 W-150.0 F10 ;
3. G40 G00 X300.0 W150.0 I40.0 K-30.0 ;

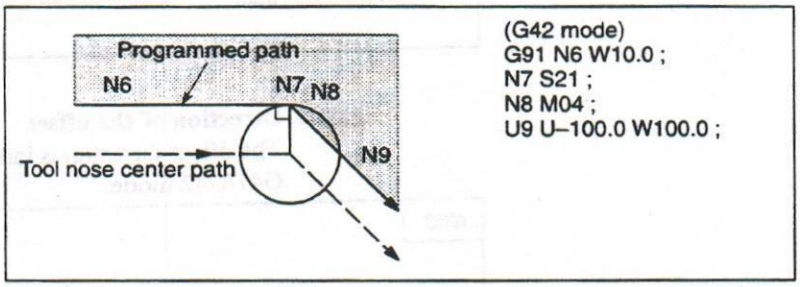
14.2.5 Notes on Tool Nose Radius Compensation

Explanations

- **Tool movement when two or more blocks without a move command should not be programmed consecutively**

1. M05 ;	M code output
2. S210 ;	S code output
3. G04 X10 ;	Dwell
4. G01 U0 ;	Feed distance of zero
5. G98 ;	G code only
6. G10 P01 X10.0 Z20.0 R0.5 Q2 ;	Offset change

If two or more of the above blocks are specified consecutively, the tool nose center comes to a position vertical to the programmed path of the preceding block at the end of the preceding block. However, if the no movement commands is **4** above, the above tool motion is attained only with one block.

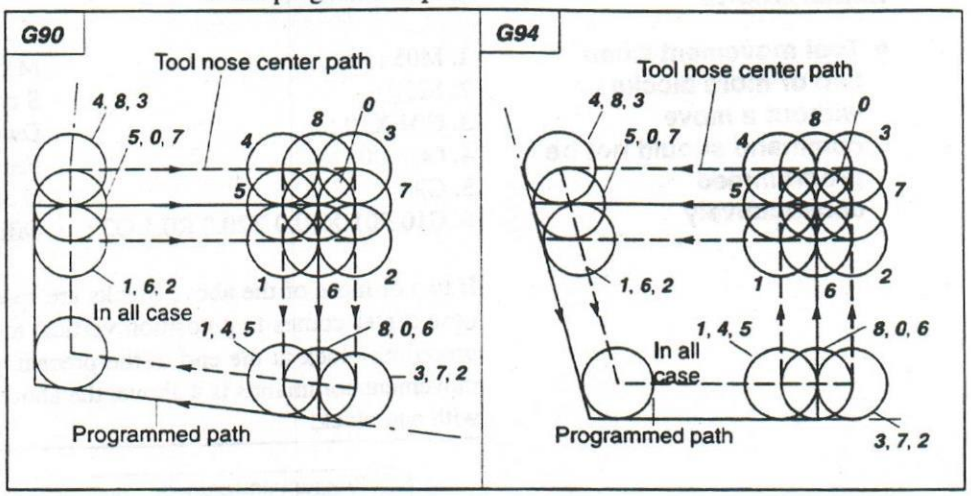


- **Tool nose radius compensation with G90 or G94**

Tool nose radius compensation with G90 (outer diameter/internal diameter cutting cycle) or G94 (end face turning cycle) is as follows, :

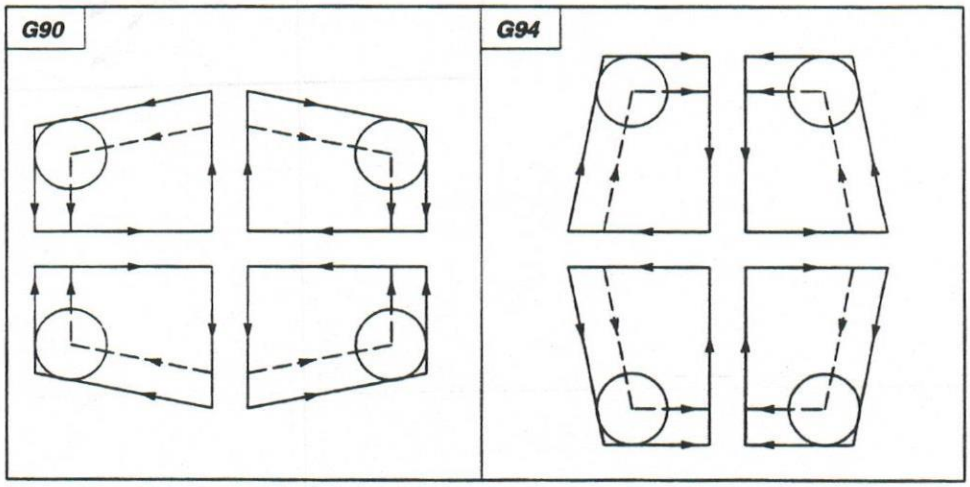
1. Motion for imaginary tool nose numbers

For each path in the cycle, the tool nose center path is generally parallel to the programmed path.



2. Direction of the offset

The offset direction is indicated in the figure below regardless of the G41/G42 mode.



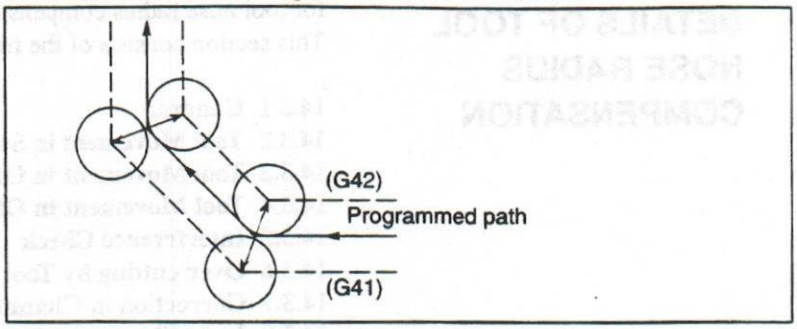
- **Tool nose radius compensation with G92**

When the following cycle is specified, the tool nose radius compensation is not performed.

G92 (Threading cycle)

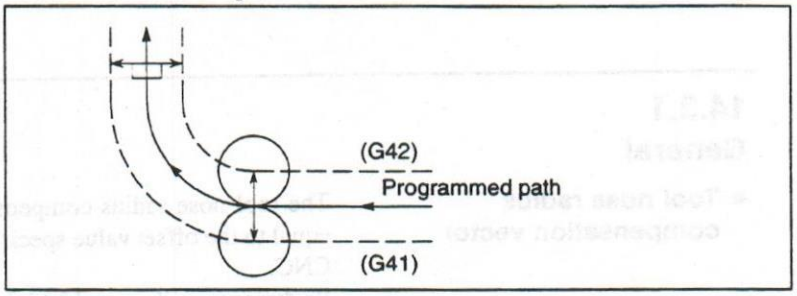
- **Tool nose radius compensation when chamfering is performed**

Movement after after compensation is shown below.



- **Tool nose radius compensation when a corner arc is inserted**

Movement after compensation is shown below.



- **Tool nose radius compensation when the block is specified from the MDI**

In this case, tool nose radius compensation is not performed.

14.3 DETAILS OF TOOL NOSE RADIUS COMPENSATION

This section provides a detailed explanation of the movement of the tool for tool nose radius compensation outlined in Section 14.2.
This section consists of the following subsections:

14.3.1 General
14.3.2 Tool Movement in Start-up
14.3.3 Tool Movement in Offset Mode
14.3.4 Tool Movement in Offset Mode Cancel
14.3.5 Interference Check
14.3.6 Over cutting by Tool Nose Radius Compensation
14.3.7 Correction in Chamfering and Corner Arc
14.3.8 Input Command from MDI
14.3.9 General Precautions for Offset Operations

14.3.1 General

- **Tool nose radius compensation vector**

The tool nose radius compensation vector is a two dimensional vector equal to the offset value specified in a T code, and the is calculated in the CNC.
Its dimension changes block by block according to tool movement.
This offset vector (simply called vector herein after) is internally created by the CNC as required for proper offsetting and to calculate a tool path with exact offset (by tool nose radius) from the programmed path.
This vector is deleted by resetting.
The vector always accompanies the tool as the tool advances.
Proper understanding of vector is essential to accurate programming.
Read the description below on how vectors are created carefully.

- **G40, G41, G42**

G40, G41 or G42 is used to delete or generate vectors.
These codes are used together with G00, G01, G02, G03 or G33 to specify a mode for tool motion (Offsetting).

G code	Function	Workpiece position
G40	Tool nose radius compensation cancel	Neither
G41	Left offset along tool path	Right
G42	Right offset along tool path	Left

G41 and G42 specify an off mode, while G40 specifies cancellation of the offset.

- **Cancel mode**

The system enters the cancel mode immediately after the power is turned on, when the RESET button on the CRT/MDI panel is pushed or a program is forced to end by executing M02 or M30. (the system may not enter the cancel mode depending on the machine tool.) In the cancel mode, the vector is set to zero, and the path of the center of tool nose coincides with the programmed, path. A program must end in cancel mode. If it ends in the offset mode, the tool cannot be positioned at the end point, and the tool stops at a location the vector length away from the end point.

• Start–up

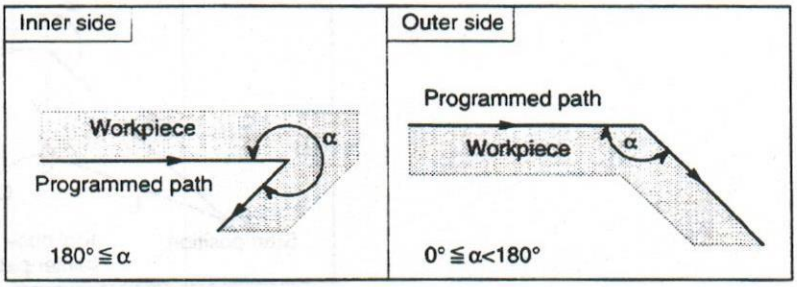
When a block which satisfies all the following conditions is executed in cancel mode, the system enters the offset mode. Control during this operation is called start–up.

- G41 or G42 is contained in the block, or has been specified to set the system enters the offset mode. Control during this operation is called start–up.
- The offset number for tool nose radius compensation is not 00.
- X or Z moves is specified in the block and the move distance is not zero.

A circular command (G02 or G03) is not allowed in start–up.
If specified, alarm (PS34) will occur. Two blocks are read in during start–up. The first block is executed, and the second block is entered into the tool nose radius compensation buffer. In the single block mode, two blocks are read and the first block is executed, then the machine stops. In subsequent operations, two blocks are read in advance, so the CNC has the block currently being executed, and the next two blocks.

• Inner side and outer side

When an angle of intersection created by tool paths specified with move commands for two blocks is over 180°, it is referred to as "inner side." When the angle is between 0° and 180°, it is referred to as "outer side."



• Meaning of symbols

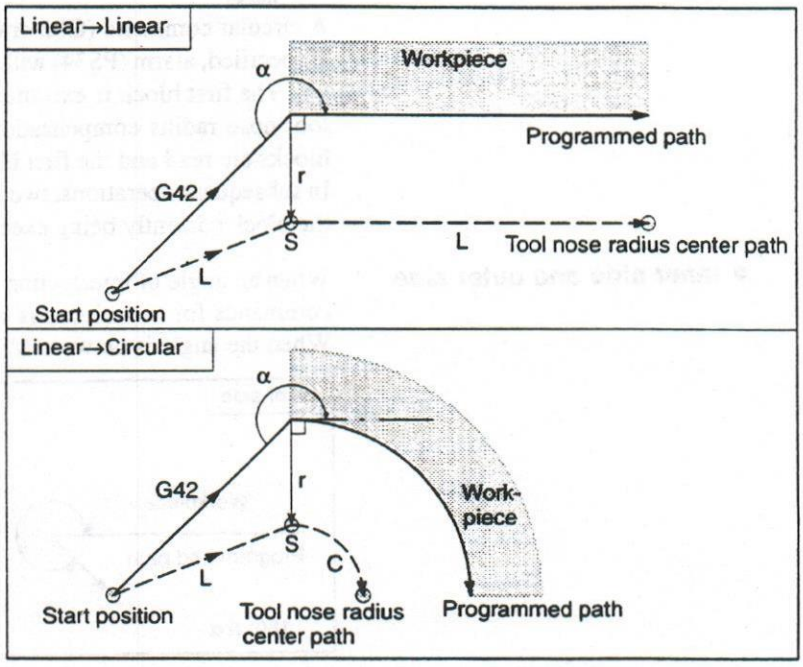
The following symbols are used in subsequent figures:
– ***S*** indicates a position at which a single block is executed once.
– ***SS*** indicates a position at which a single block is executed twice.
– ***SSS*** indicates a position at which a single block is executed three times.
– ***L*** indicates that the tool moves along a straight line.
– ***C*** indicates that the tool moves along an arc.
– ***r*** indicates the tool nose radius compensation value.
– An intersection is a position at which the programmed paths of two blocks intersect with each other after they are shifted by ***r***.
– indicates ○ the center of the tool nose radius.

14.3.2 Tool Movement in Start–up

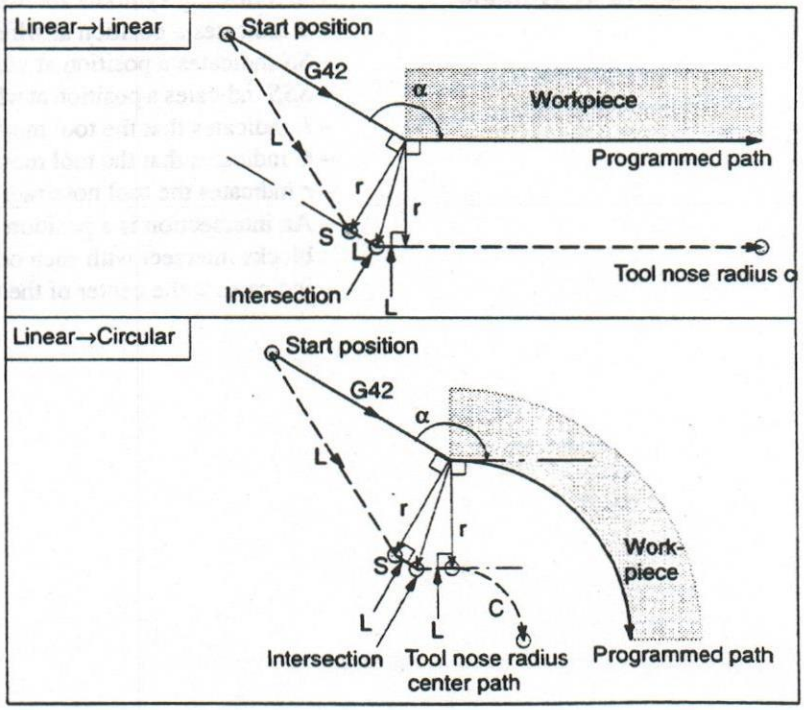
When the offset cancel mode is changed to offset mode, the tool moves as illustrated below (start–up):

Explanations

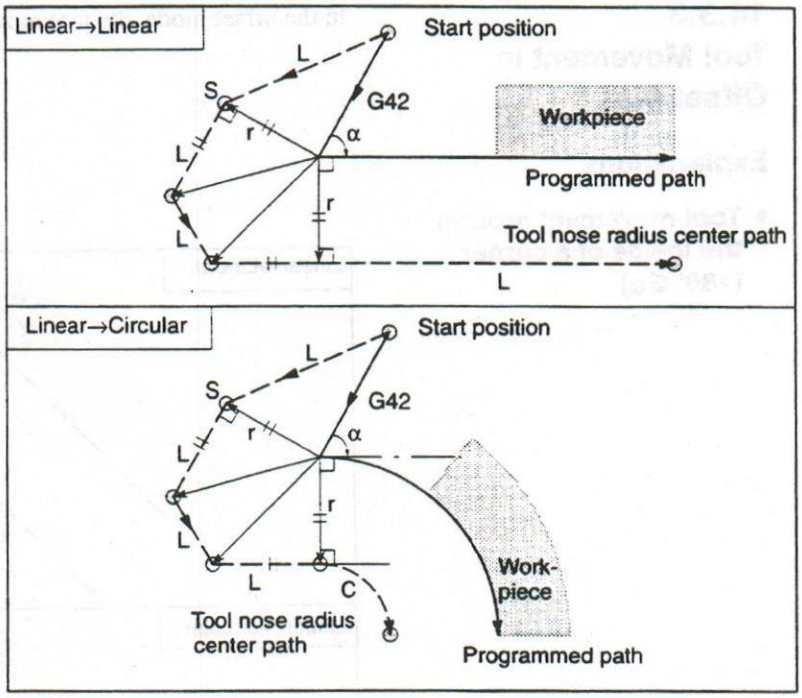
- **Tool movement around an inner side of a corner (180° ≦ α)**



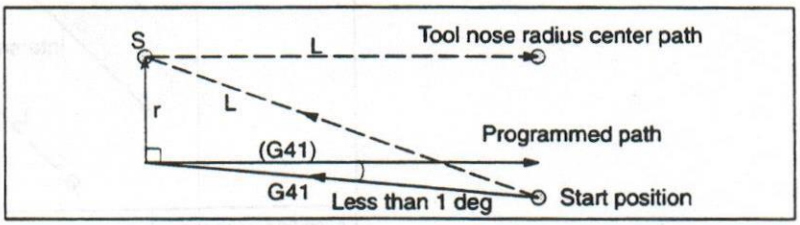
- **Tool movement around the outside of a corner at an obtuse angle (90° ≦ α<180°)**



- **Tool movement around the outside of an acute angle (α<90°)**

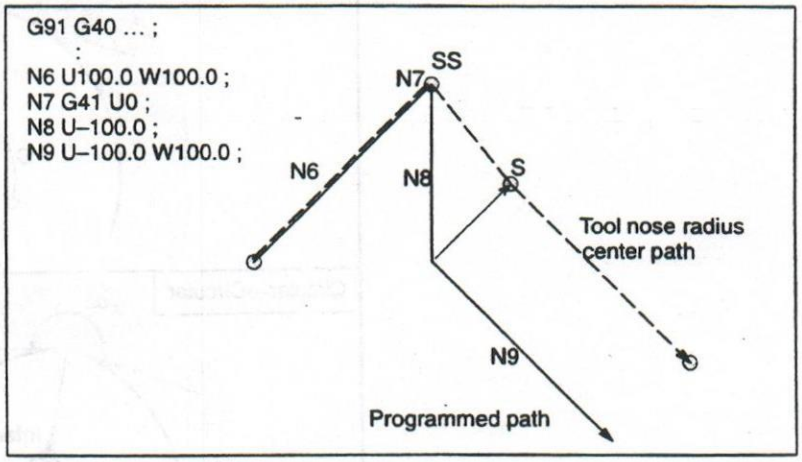


- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



- **A block without tool movement specified at start–up**

If the command is specified at start–up, the offset vector is not created.



Notes

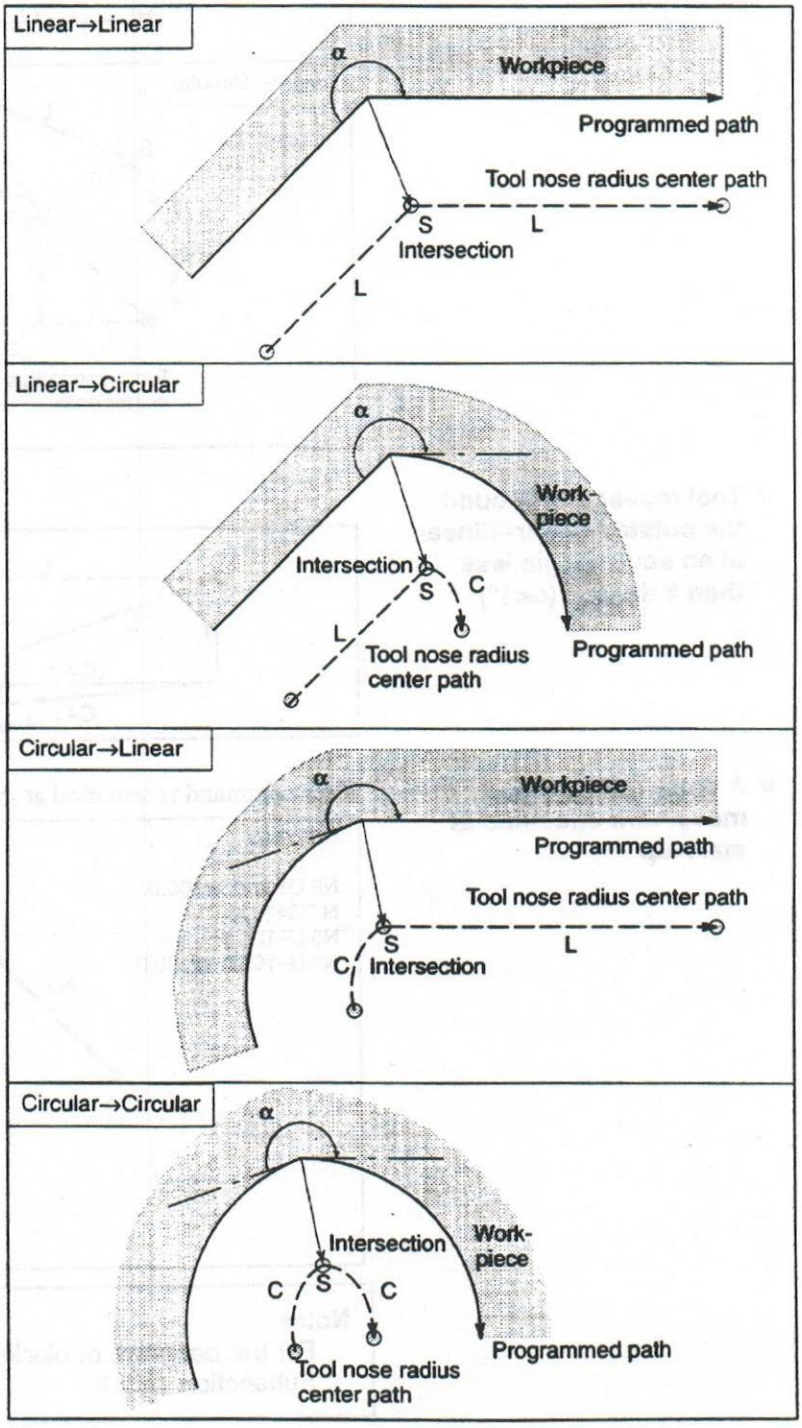
For the definition of blocks that do not move the tool, see Subsection 14.3.3.

14.3.3 Tool Movement in Offset Mode

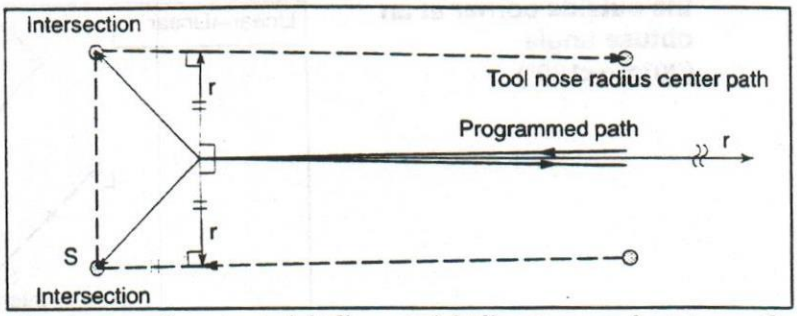
In the offset mode, the tool moves as illustrated below:

Explanations

- **Tool movement around the inside of a corner ($180° \leqq \alpha$)**

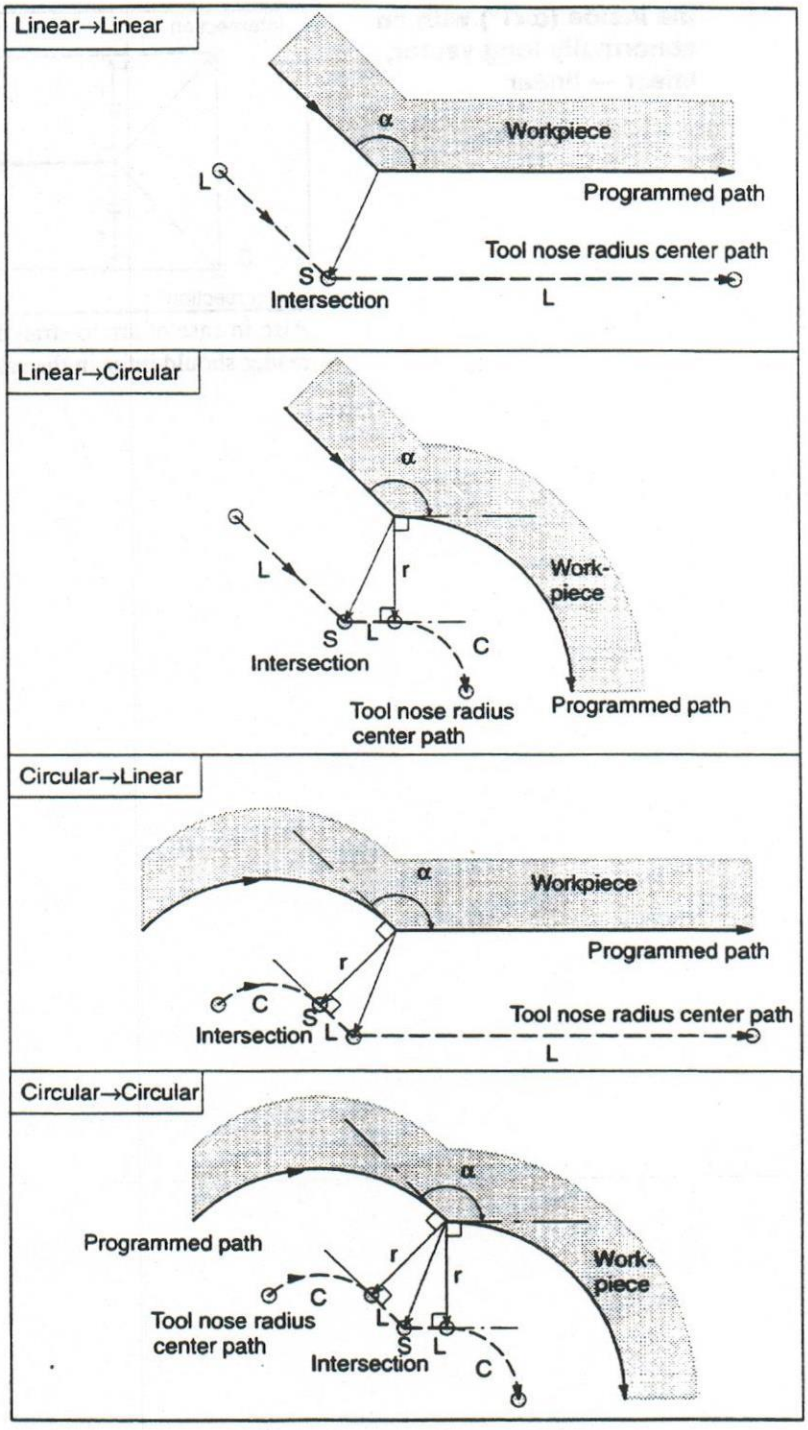


- **Tool movement around the inside (α<1°) with an abnormally long vector, linear → linear**

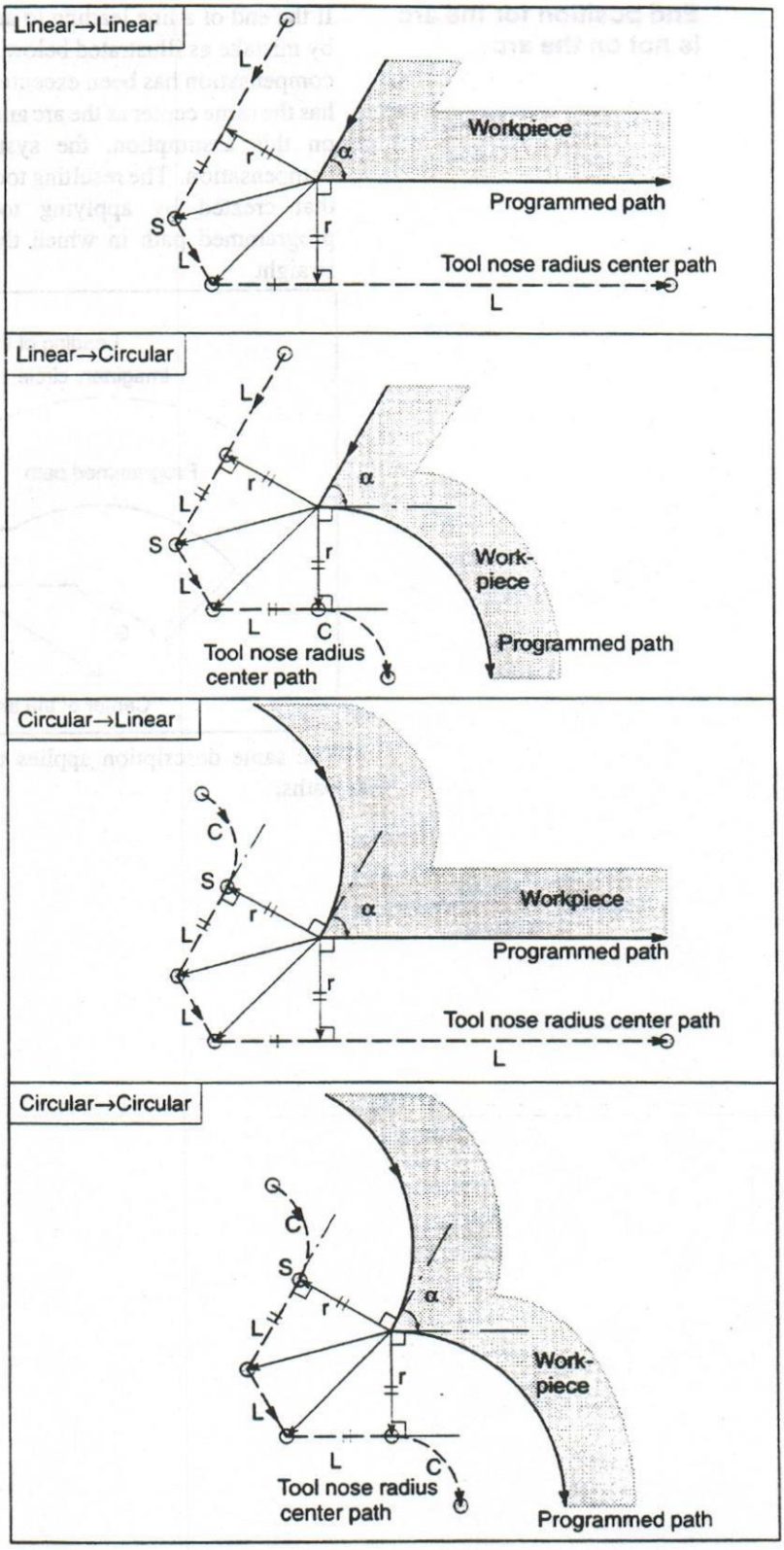


Also in case of arc to straight line, straight line to arc and arc to arc, the reader should infer in the same procedure.

- **Tool movement around the outside corner at an obtuse angle ($90° \leqq \alpha < 180°$)**



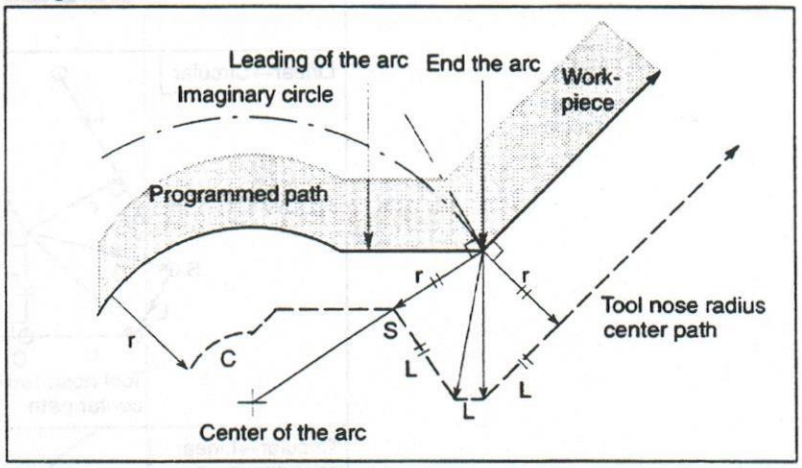
- **Tool movement around the outside corner at an acute angle (α<90°)**



- **When it is exceptional**

End position for the arc is not on the arc

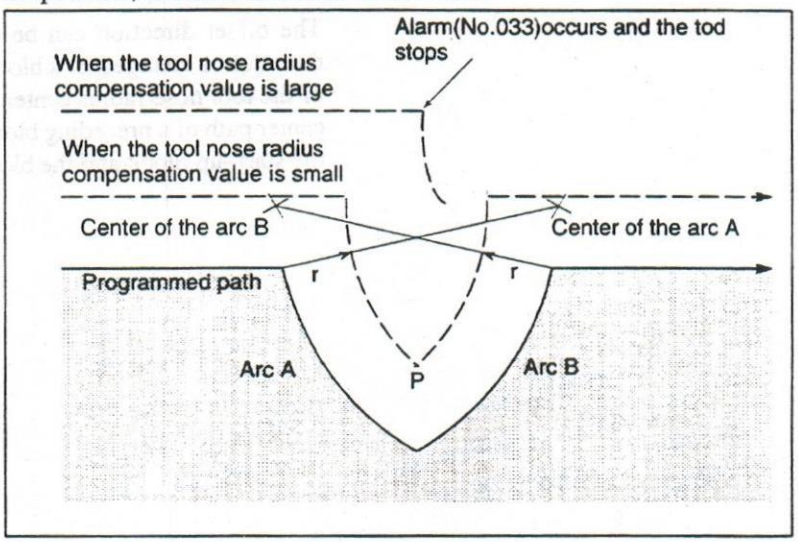
If the end of a line leading to an arc is programmed as the end of the arc by mistake as illustrated below, the system assumes that tool nose radius compensation has been executed with respect to an imaginary circle that has the same center as the arc and passes the specified end position. Based on this assumption, the system creates a vector and carries out compensation. The resulting tool nose radius center path is different from that created by applying tool nose radius compensation to the programmed path in which the line leading to the arc is considered straight.



The same description applies to tool movement between two circular paths.

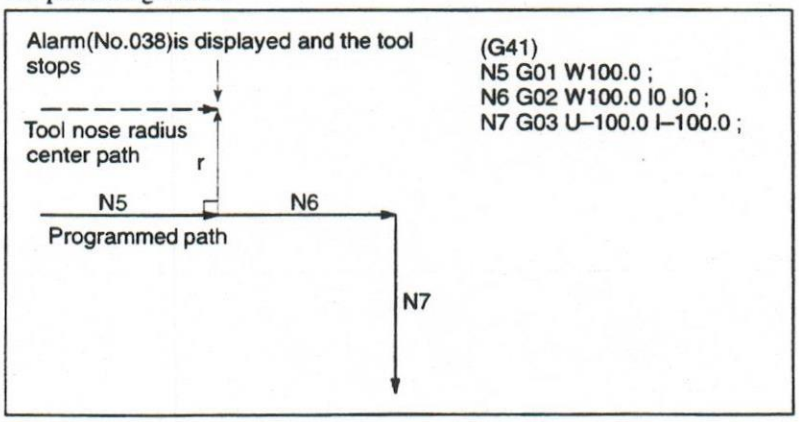
There is no inner intersection

If the tool nose radius compensation value is sufficiently small, the two circular Tool nose radius center paths made after compensation intersect at a position (P). Intersection P may not occur if an excessively large value is specified for tool nose radius compensation. When this is predicted, alarm 33 occurs at the end of the previous block and the tool is stopped. In the example shown below, Tool nose radius center paths along arcs A and B intersect at P when a sufficiently small value is specified for tool nose radius compensation. If an excessively large value is specified, this intersection does not occur.



The center of the arc is identical with the start position or the end position

If the center of the arc is identical with the start position or end point, alarm (No. 038) is displayed, and the tool will stop at the end position of the preceding block.



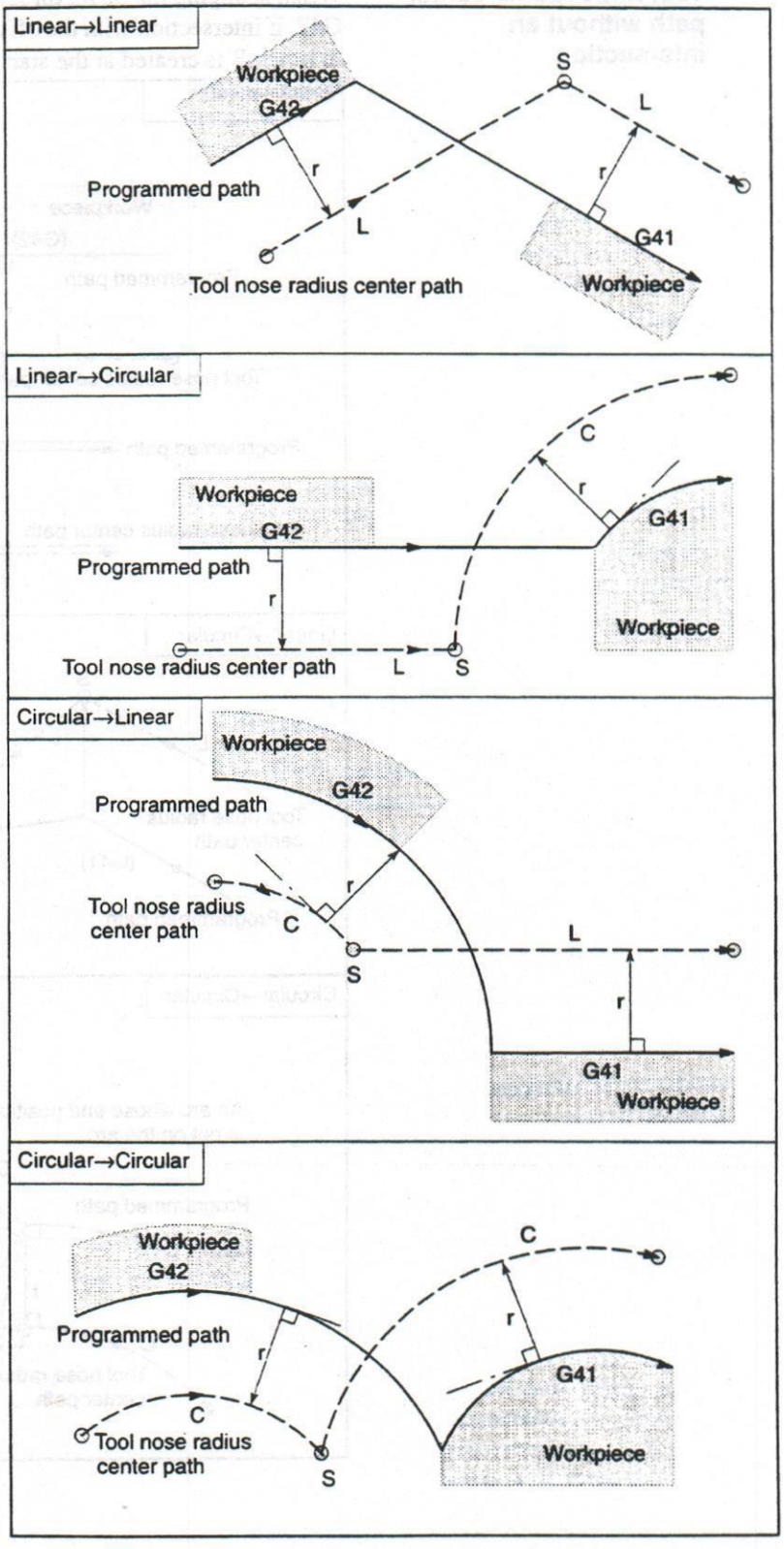
• Change in the offset direction in the offset mode

The offset direction is decided by G codes (G41 and G42) for tool nose radius and the sign of tool nose radius compensation value as follows.

G code \ Sign of offset value	+	–
G41	Left side offset	Right side offset
G42	Right side offset	Left side offset

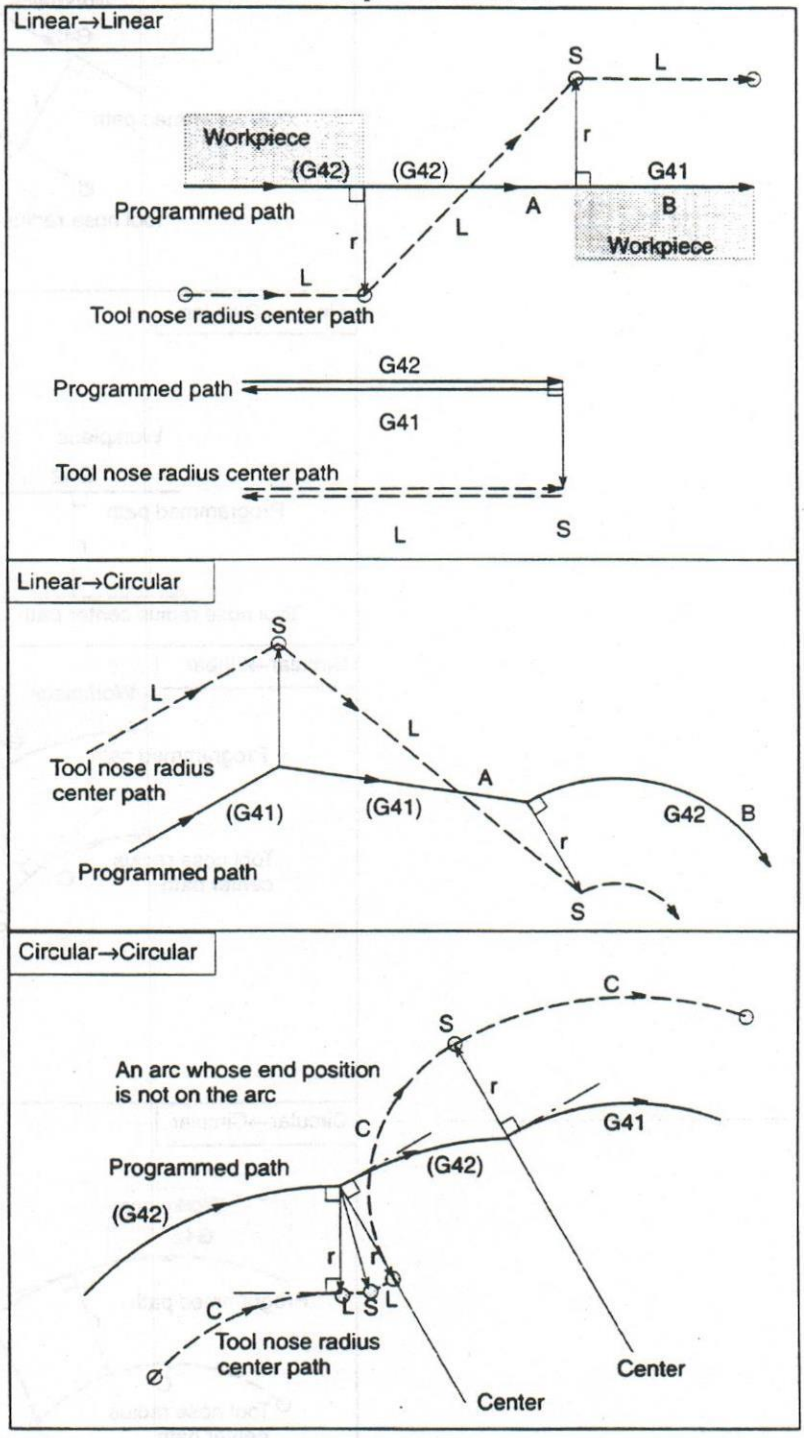
The offset direction can be changed in the offset mode. If the offset direction is changed in a block, a vector is generated at the intersection of the tool nose radius center path of that block and the tool nose radius center path of a preceding block. However, the change is not available in the start–up block and the block following it.

Tool nose radius center path with an intersection



Tool nose radius center path without an intersection

When changing the offset direction in block A to block B using G41 and G42, if intersection with the offset path is not required, the vector normal to block B is created at the start point of block B.

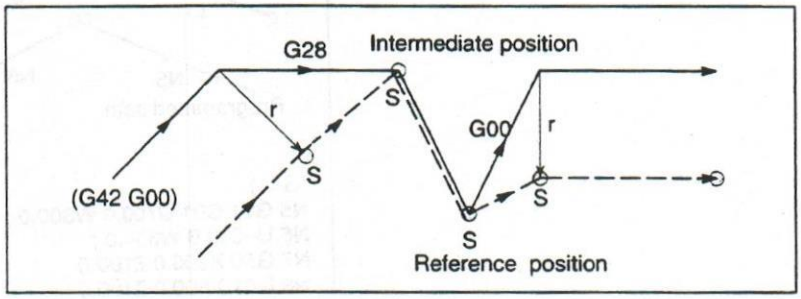


- **Temporary tool nose radius compensation cancel**

If the following command is specified in the offset mode, the offset mode is temporarily canceled then automatically restored. The offset mode can be canceled and started as described in Subsections 14.3.2 and 14.3.4.

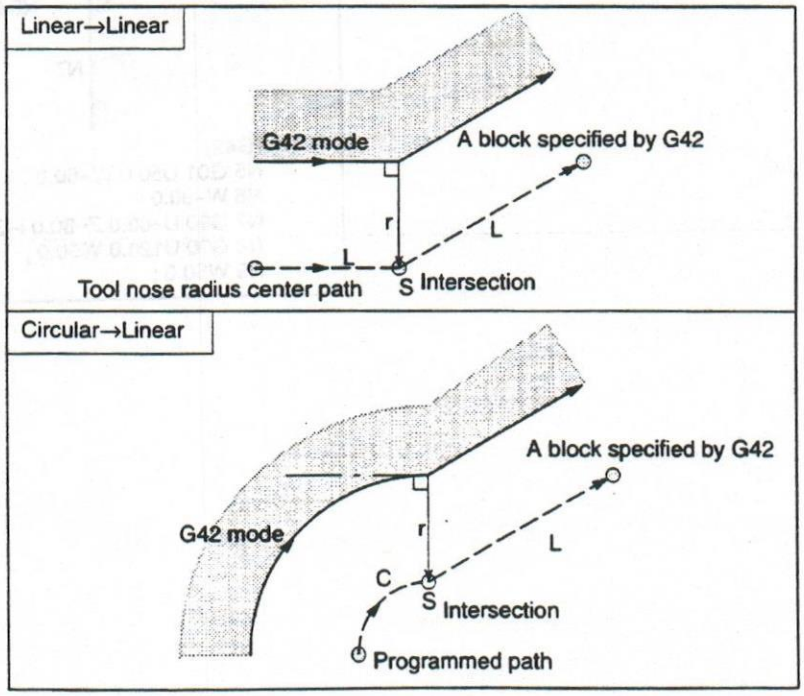
Specifying G28 (automatic return to the reference position) in the offset mode

If G28 is specified in the offset mode, the offset mode is canceled at an intermediate position. If the vector still remains after the tool is returned to the reference position, the components of the vector are reset to zero with respect to each axis along which reference position return has been made.



- **Tool nose radius compensation G code in the offset mode**

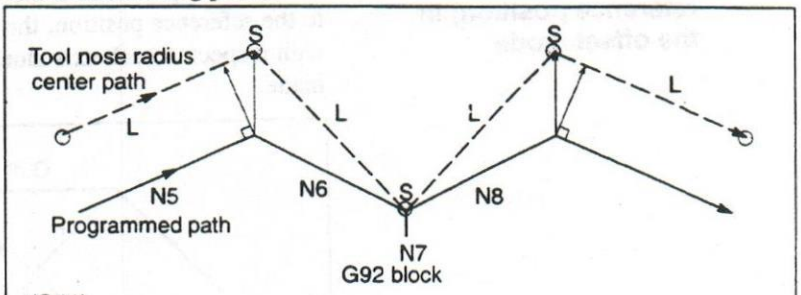
The offset vector can be set to form a right angle to the moving direction in the previous block, irrespective of machining inner or outer side, by commanding the tool nose radius compensation G code (G41, G42) in the offset mode, independently. If this code is specified in a circular command, correct circular motion will not be obtained.
When the direction of offset is expected to be changed by the command of tool nose radius compensation G code (G41, G42), refer to "Change in the offset direction in the offset mode" in Subsec.14.3.3.



- **Command cancelling the offset vector temporality**

During offset mode, if G92 (absolute zero point programming) is commanded,the offset vector is temporarily cancelled and thereafter offset mode is automatically restored.
In this case, without movement of offset cancel, the tool moves directly from the intersecting point to the commanded point where offset vector is canceled. Also when restored to offset mode, the tool moves directly to the intersecting point.

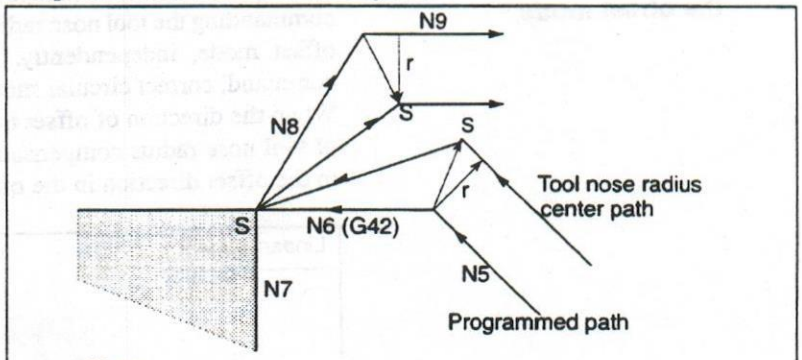
- **Workpiece coordinate system setting (G50)**




```
(G41)
N5 G91 G01 U700.0 W300.0 ;
N6 U–600.0 W300.0 ;
N7 G50 X200.0 Z100.0 ;
N8 G01 X800.0 Z400.0 ;
```

- **Canned cycles (G90, G92, G94)**

See Sections 14.2.5 (G90, G92, G94) for the tool nose radius compensation is related canned cycles.




```
(G42)
N5 G01 U50.0 W–60.0 ;
N6 W–80.0 ;
N7 G90 U–60.0 Z–80.0 I–30.0 ;
N8 G00 U120.0 W50.0 ;
N9 W50.0 ;
```

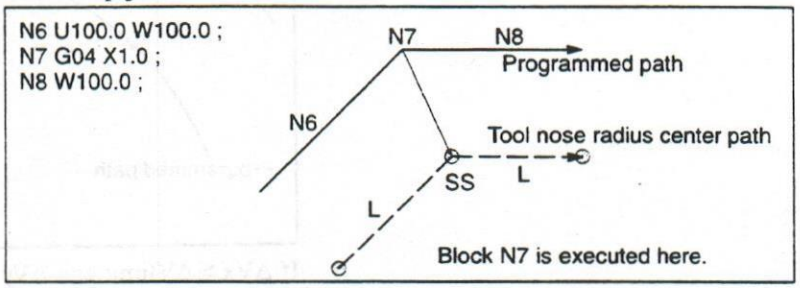
• A block without tool movement

The following blocks have no tool movement. In these blocks, the tool will not move even if tool nose radius compensation is effected.

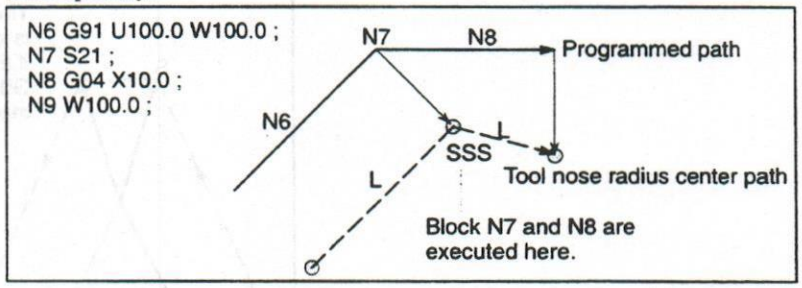
1. M05 ;	M code output	Commands 1 to 6 are of no movement.
2. S21 ;	S code output	
3. G04 X10.0 ;	Dwell	
4. G10 P01 X10 Z20 R10.0 ;	tool nose radius compensation value setting	
5. G98 ;	G code only	
6. U0 ;	Move distance is zero.	

A block without tool movement specified in offset mode

When a single block without tool movement is commanded in the offset mode, the vector and Tool nose radius center path are the same as those when the block is not commanded. This block is executed at the single block stop point.

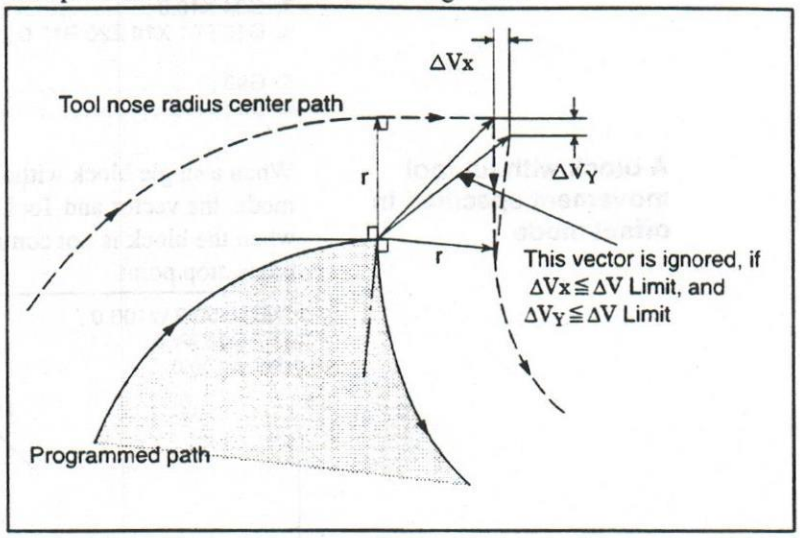


However, when the move distance is zero, even if the block is commanded singly, tool motion becomes the same as that when more than one block of without tool movement are commanded, which will be described subsequently.

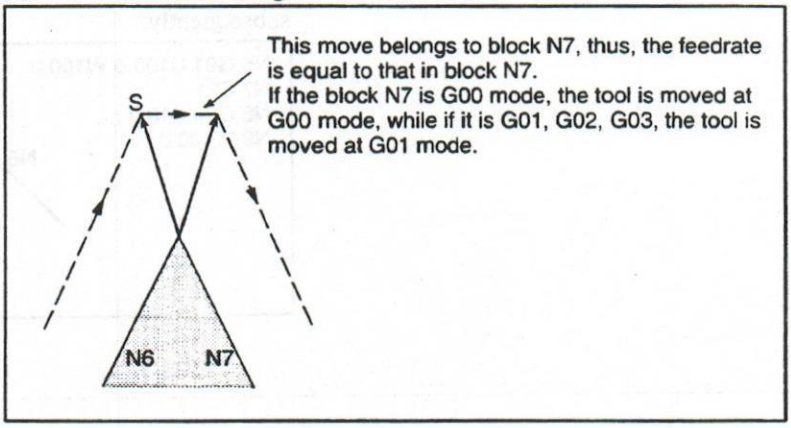


- **Corner movement**

When two or more vectors are produced at the end of a block, the tool moves linearly from one vector to another. This movement is called the corner movement.
If these vectors almost coincide with each other, the corner movement isn't performed and the latter vector is ignored.



If $\Delta Vx \leqq \Delta Vlimit$ and $\Delta Vy \leqq \Delta Vlimit$, the latter vector is ignored. The $\Delta Vlimit$ is set in advance by parameter (No. 5010).
If these vectors do not coincide, a move is generated to turn around the corner. This move belongs to the latter block.


This move belongs to block N7, thus, the feedrate is equal to that in block N7.
If the block N7 is G00 mode, the tool is moved at G00 mode, while if it is G01, G02, G03, the tool is moved at G01 mode.
S
N6
N7

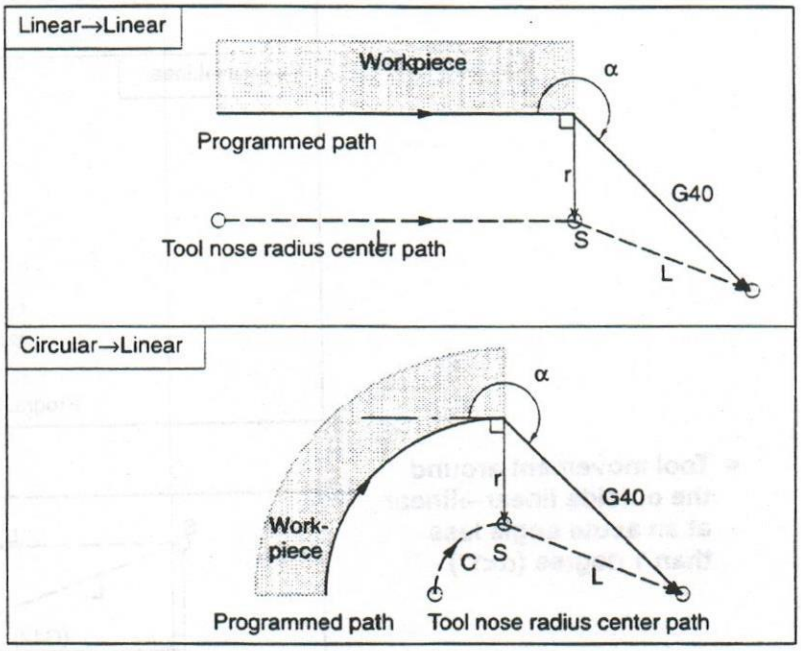
- **Interruption of manual operation**

For manual operation during the tool nose radius compensation, refer to Section III–3.5, " ManualAbsolute ON and OFF."

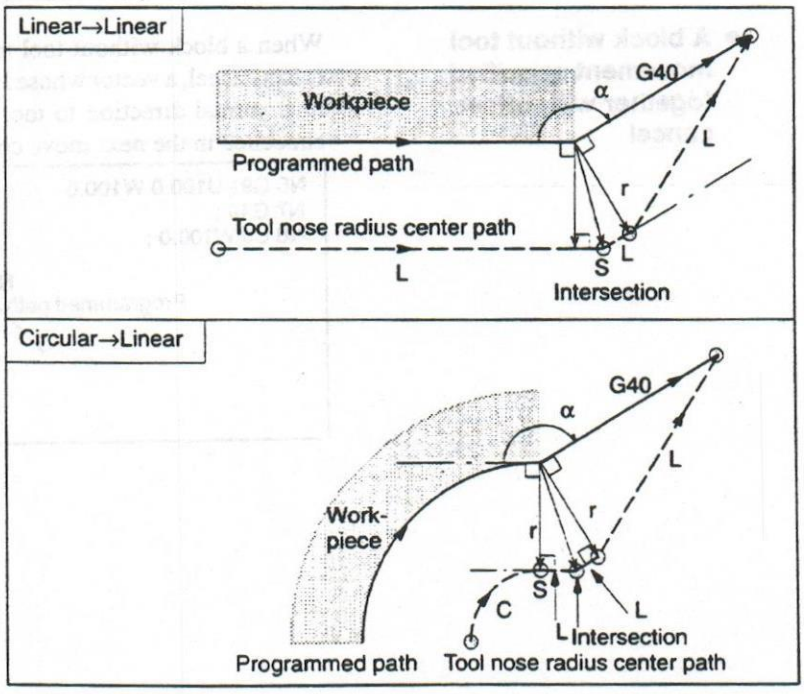
14.3.4 Tool Movement in Offset Mode Cancel

Explanations

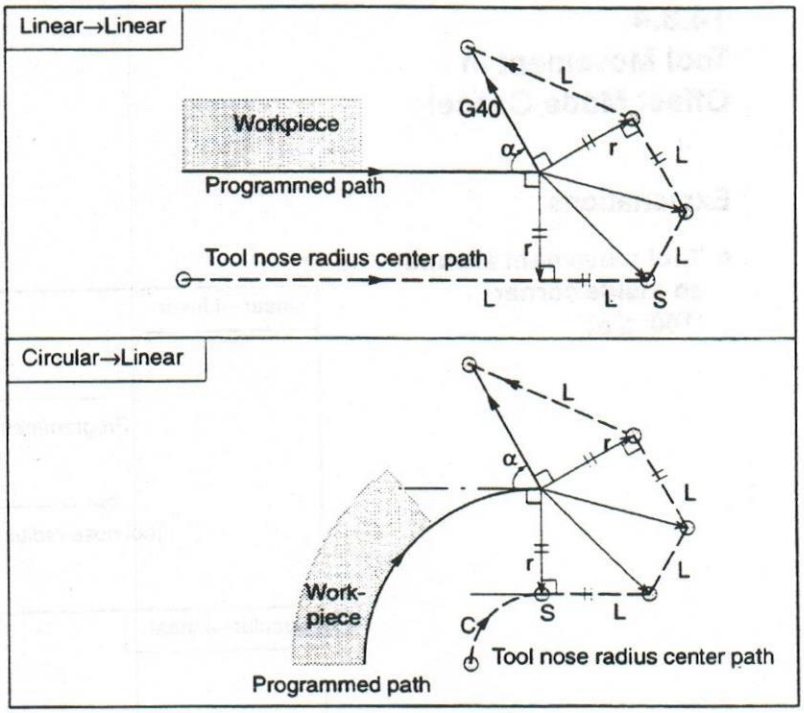
- **Tool movement around an inside corner ($180° \leqq \alpha$)**



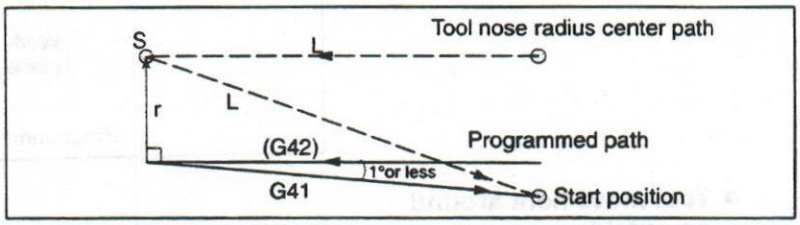
- **Tool movement around an outside corner at an obtuse angle ($90° \leqq \alpha < 180°$)**



- **Tool movement around an outside corner at an acute angle (α<90°)**

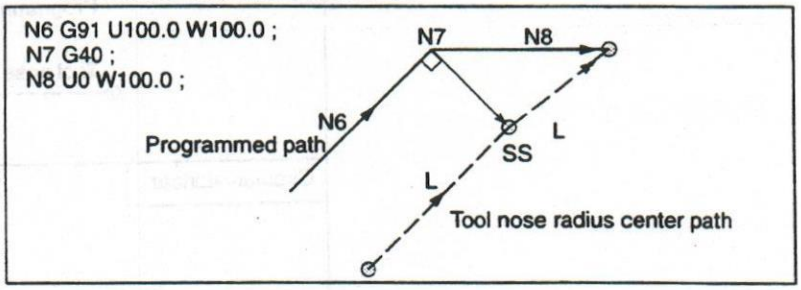


- **Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°)**



- **A block without tool movement specified together with offset cancel**

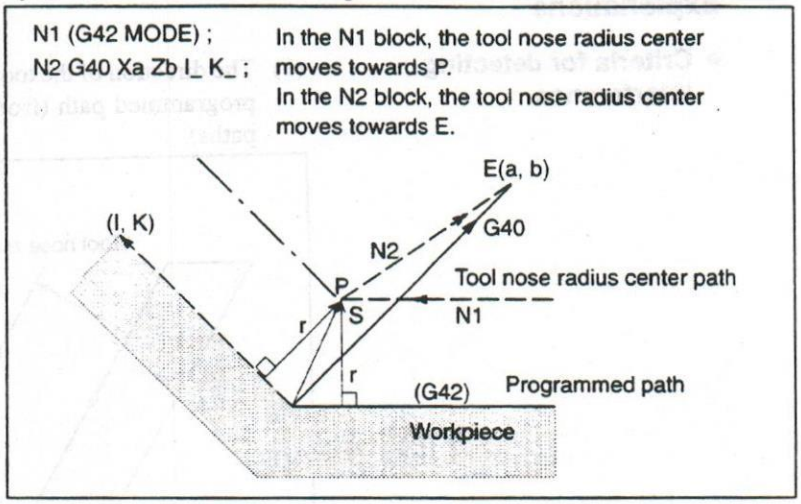
When a block without tool movement is commanded together with an offset cancel, a vector whose length is equal to the offset value is produced in a normal direction to tool motion in the earlier block, the vector is cancelled in the next move command.



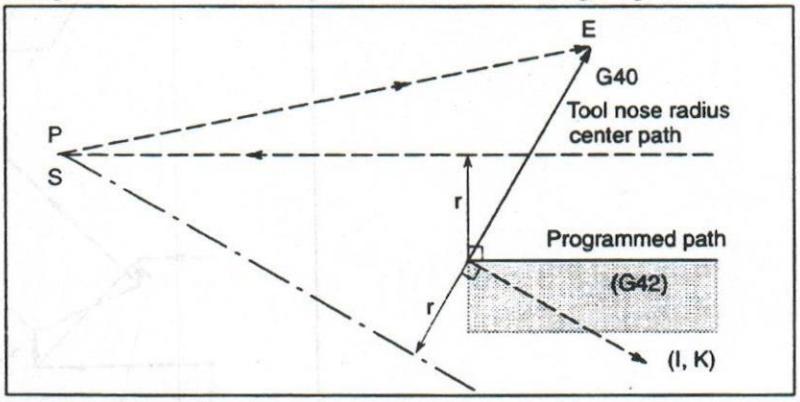
- **Block containing G40 and I_J_K_**

The previous block contains G41 or G42

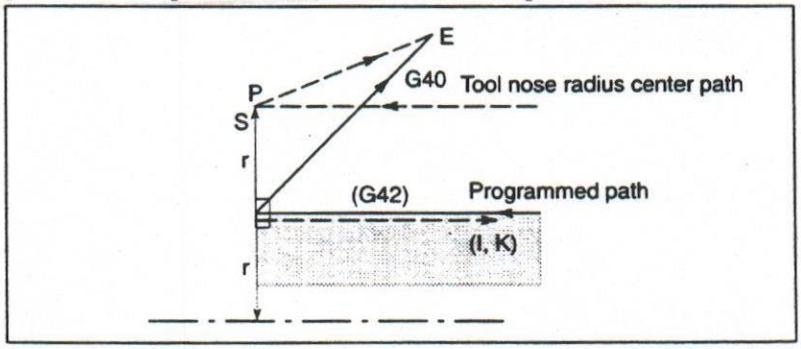
If a G41 or G42 block precedes a block in which G40 and I_, K_ are specified, the system assumes that the path is programmed as a path from the end position determined by the former block to a vector determined by (I, K). The direction of compensation in the former block is inherited.



In this case, note that the CNC obtains an intersection of the tool path irrespective of whether inner or outer side machining is specified



When an intersection is not obtainable, the tool comes to the normal position to the previous block at the end of the previous block.



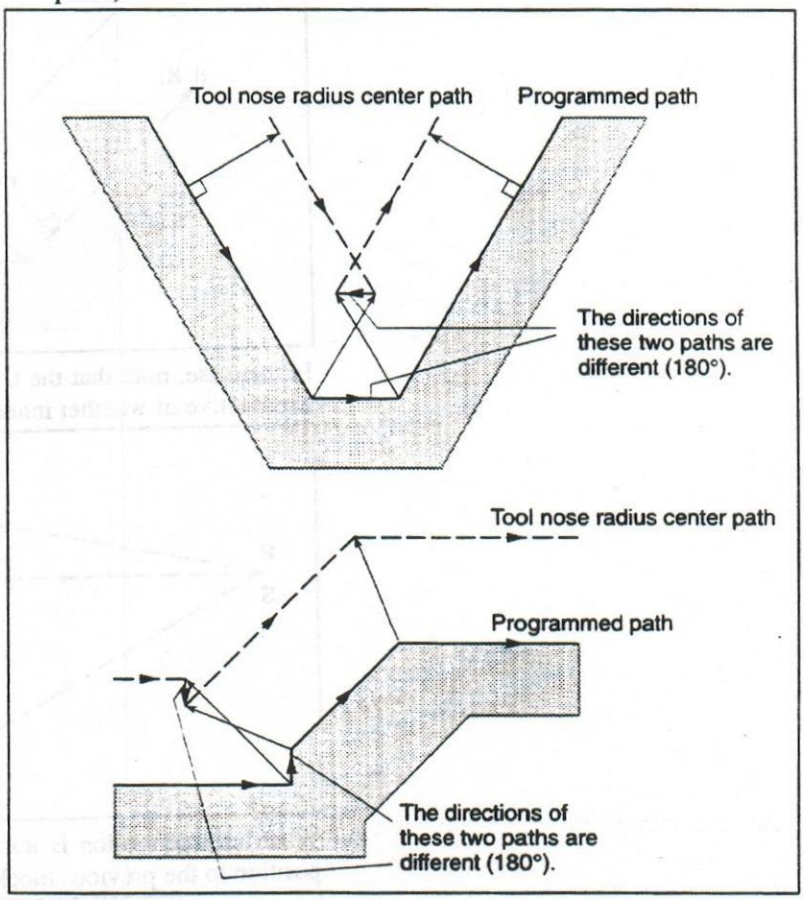
14.3.5 Interference Check

Tool overcutting is called interference. The interference check function checks for tool overcutting in advance. However, all interference cannot be checked by this function. The interference check is performed even if overcutting does not occur.

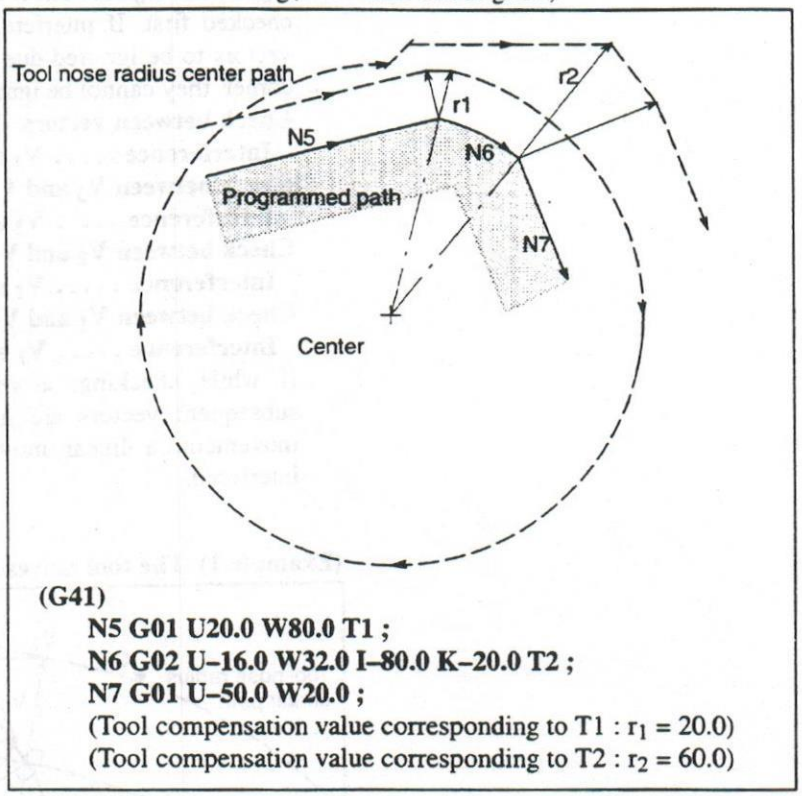
Explanations

- **Criteria for detecting interference**

(1) The direction of the tool nose radius path is different from that of the programmed path (from 90 degrees to 270 degrees between these paths).


Tool nose radius center path
Programmed path
The directions of these two paths are different (180°).
Tool nose radius center path
Programmed path
The directions of these two paths are different (180°).

(2) In addition to the condition (1), the angle between the start point and end point on the Tool nose radius center path is quite different from that between the start point and end point on the programmed path in circular machining(more than 180 degrees).



(G41)

N5 G01 U20.0 W80.0 T1 ;
N6 G02 U–16.0 W32.0 I–80.0 K–20.0 T2 ;
N7 G01 U–50.0 W20.0 ;
(Tool compensation value corresponding to T1 : r_1 = 20.0)
(Tool compensation value corresponding to T2 : r_2 = 60.0)

In the above example, the arc in block N6 is placed in the one quadrant. But after tool nose radius compensation, the arc is placed in the four quadrants.

- **Correction of interference in advance**

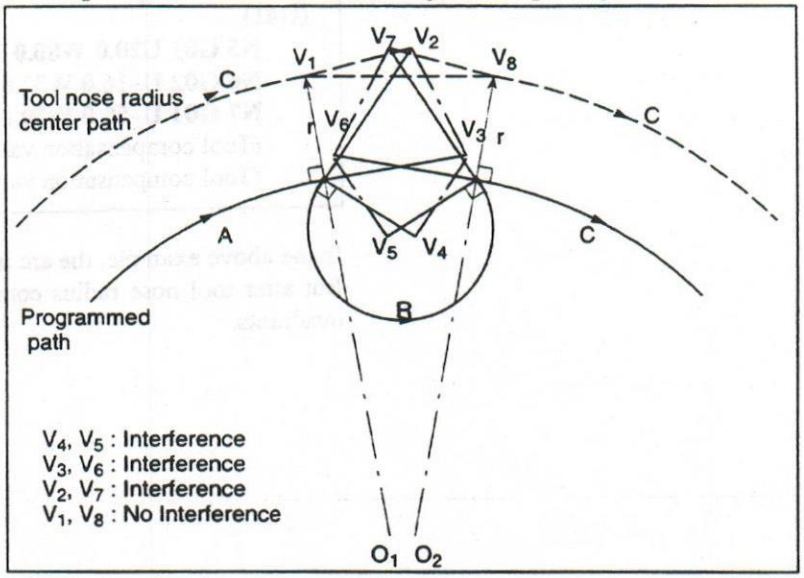
(1) Removal of the vector causing the interference

When tool nose radius compensation is performed for blocks A, B and C and vectors V_1, V_2, V_3 and V_4 between blocks A and B, and V_5, V_6, V_7 and V_8 between B and C are produced, the nearest vectors are checked first. If interference occurs, they are ignored. But if the vectors to be ignored due to interference are the last vectors at the corner, they cannot be ignored.

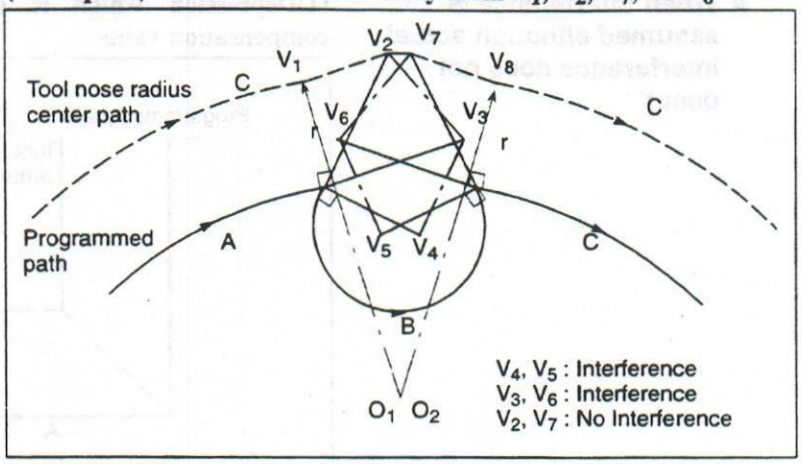
Check between vectors V_4 and V_5
Interference ..... V_4 and V_5 are ignored.
Check between V_3 and V_6
Interference ..... V_3 and V_6 are ignored
Check between V_2 and V_7
Interference ..... V_2 and V_7 are Ignored
Check between V_1 and V_8
Interference ..... V_1 and V_8 are cannot be ignored

If while checking, a vector without interference is detected, subsequent vectors are not checked. If block B is a circular movement, a linear movement is produced if the vectors are interfered.

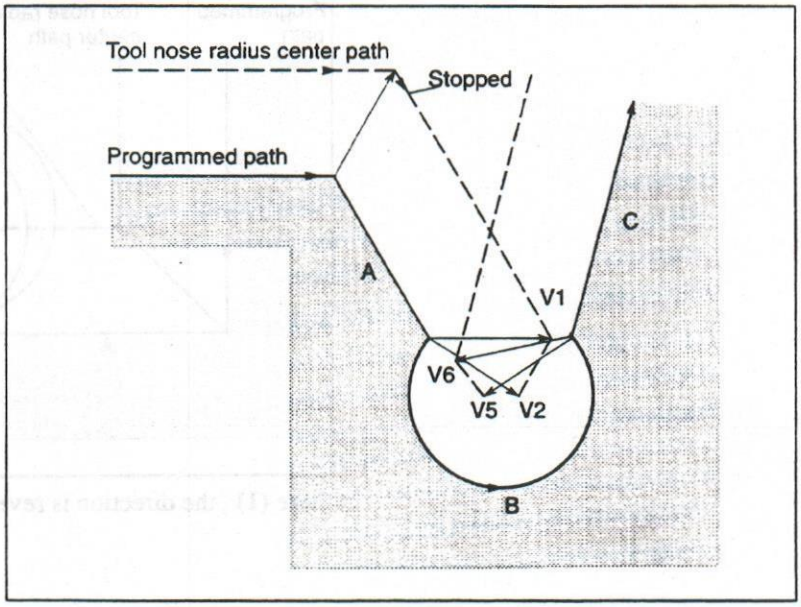
(Example 1) The tool moves linearly from V_1 to V_8



(Example 2) The tool moves linearly from V_1, V_2, V_7, to V_8



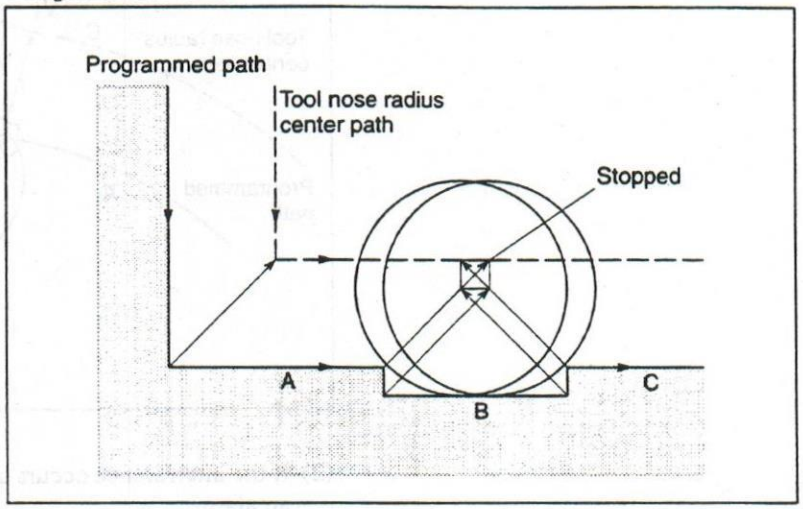
(2) If the interference occurs after correction (1), the tool is stopped with an alarm.
If the interference occurs after correction (1) or if there are only one pair of vectors from the beginning of checking and the vectors interfere, the alarm (No.41) is displayed and the tool is stopped immediately after execution of the preceding block. If the block is executed by the single block operation, the tool is stopped at the end of the block.



After ignoring vectors V_2 and V_5 because of interference, interference also occurs between vectors V_1 and V_6. The alarm is displayed and the tool is stopped.

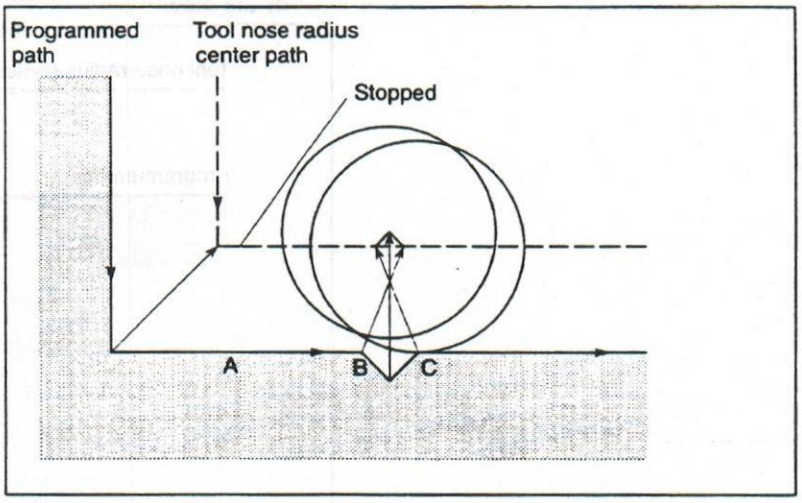
- **When interference is assumed although actual interference does not occur**

(1)Depression which is smaller than the tool nose radius compensation value



There is no actual interference, but since the direction programmed in block B is opposite to that of the path after tool nose radius compensation the tool stops and an alarm(No.041) is displayed.

(2)Groove which is smaller than the tool nose radius compensation value



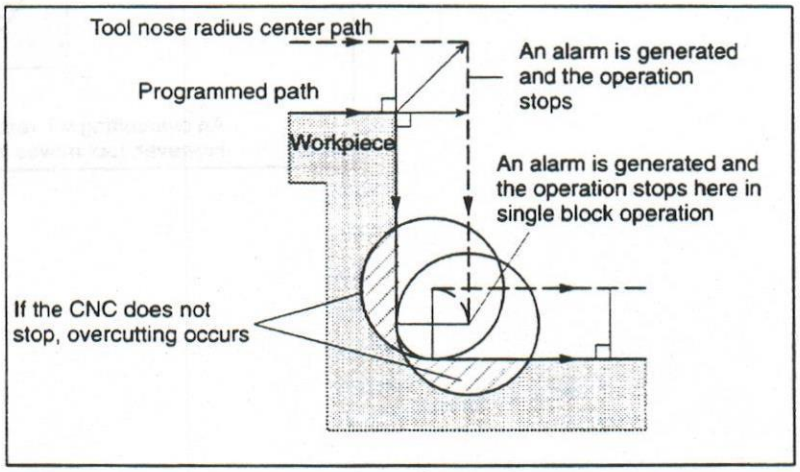
Like (1) , the direction is reverse in block B.

14.3.6 Overcutting by Tool Nose Radius Compensation

Explanations

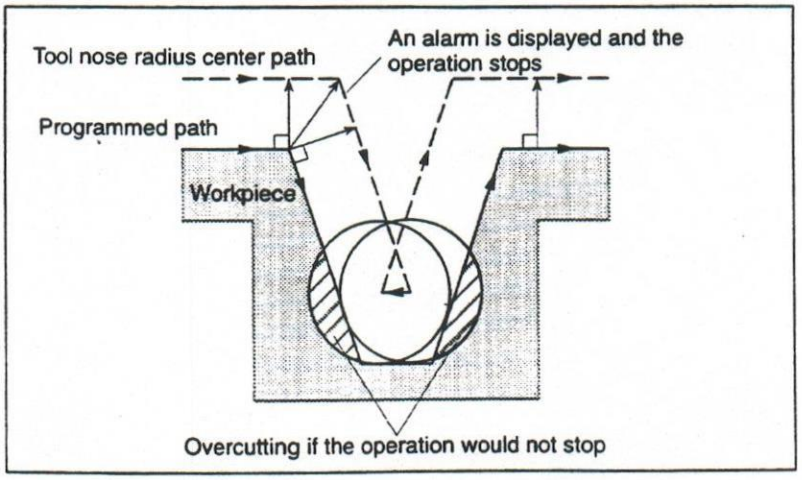
- **Machining an inside corner at a radius smaller than the tool nose radius**

When the radius of a corner is smaller than the cutter radius, because the inner offsetting of the cutter will result in overcuttings, an alarm is displayed and the CNC stops at the start of the block. In single block operation, the overcutting is generated because the tool is stopped after the block execution.


Tool nose radius center path
Programmed path
Workpiece
An alarm is generated and the operation stops
An alarm is generated and the operation stops here in single block operation
If the CNC does not stop, overcutting occurs

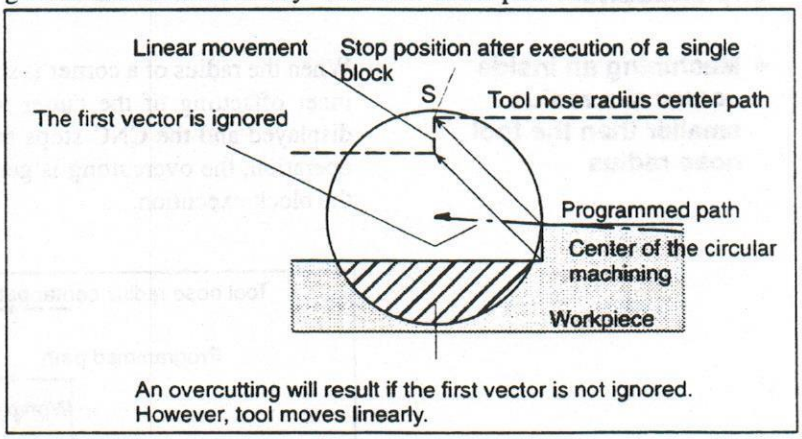
- **Machining a groove smaller than the tool nose radius**

Since the tool nose radius compensation forces the path of the center of the tool to move in the reverse of the programmed direction, overcutting will result. In this case an alarm is displayed and the CNC stops at the start of the block.


Tool nose radius center path
An alarm is displayed and the operation stops
Programmed path
Workpiece
Overcutting if the operation would not stop

- **Machining a step smaller than the tool nose radius**

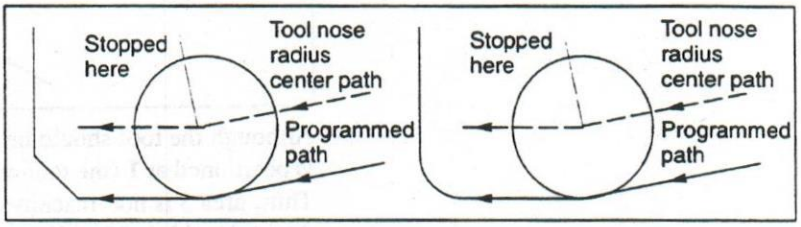
When machining of the step is commanded by circular machining in the case of a program containing a step smaller than the tool nose radius, the path of the center of tool with the ordinary offset becomes reverse to the programmed direction. In this case, the first vector is ignored, and the tool moves linearly to the second vector position. The single block operation is stopped at this point. If the machining is not in the single block mode, the cycle operation is continued. If the step is of linear, no alarm will be generated and cut correctly. However uncut part will remain.



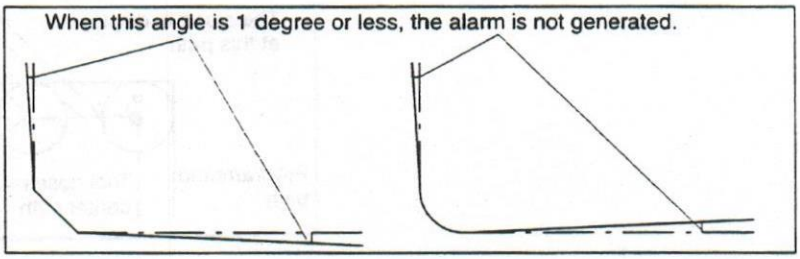
An overcutting will result if the first vector is not ignored.
However, tool moves linearly.

14.3.7 Correction in Chamfering and Corner Arcs

In chamfering or corner arcs, tool nose radius compensation only be performed when an ordinary intersection exists at the corner.
In offset cancel mode, a start–up block or when exchanging the offset direction, compensation cannot be performed, an alarm (No.39) is displayed and the tool is stopped.
In inner chamfering or inner corner arcs, if the chamfering value or corner arc value is smaller than the tool nose radius value, the tool is stopped with an alarm (No.39) since overcutting will occur.

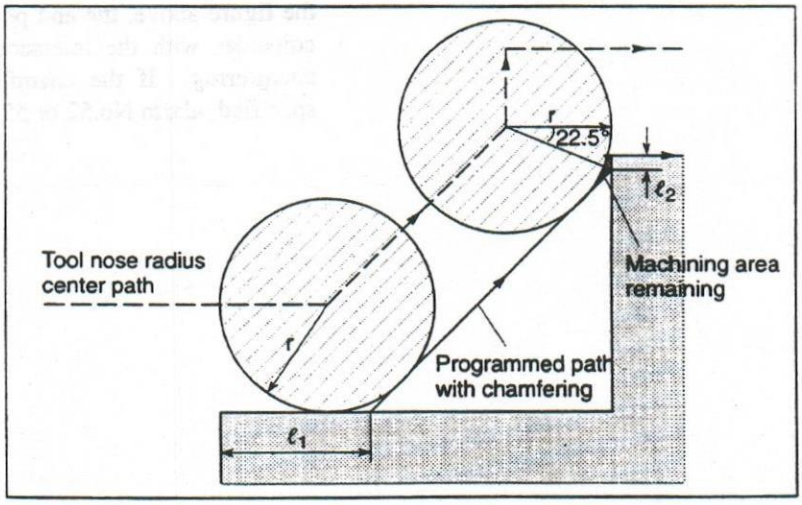


The valid inclination angle of the programmed path in the blocks before and after the corner is 1 degree or less so that the alarm (No.52, 54) generated by the calculating error of tool nose radius compensation does not occur.

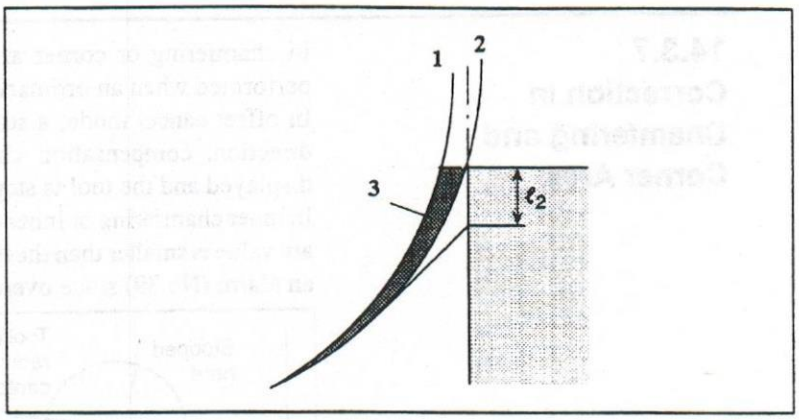


• When machining area remains or an alarm is generated

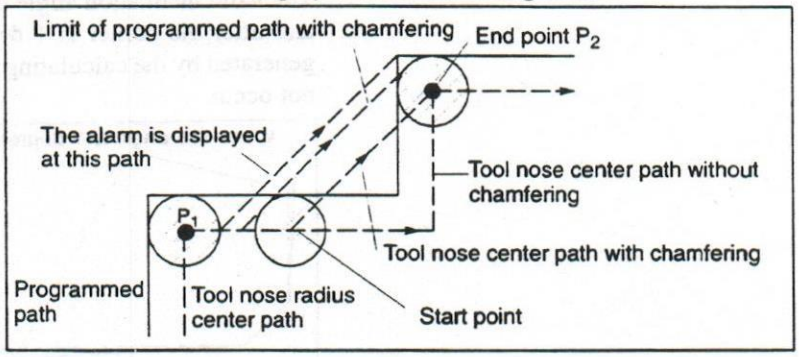
The following example shows a machining area which cannot be cut sufficiently.



In inner chamfering, if the portion of the programmed path that is not a part of the chamfering (in the above figure ℓ_1 or ℓ_2) is in following range, insufficiently cut are will exist.
$0 \leqq \ell_1$ or $\ell_2 < r \cdot \tan 22.5°$ (r : too nose radius)
Enlarged view on the remaining machining area



Although the tool should be positioned at **2** in the above figure, the tool is positioned at **1** (the tool nose is tangent to line L).
Thus, area **3** is not machined.
Alarm No.52 or 55 is displayed in the following cases :


Limit of programmed path with chamfering
End point P_2
The alarm is displayed at this path
Tool nose center path without chamfering
P_1
Tool nose center path with chamfering
Programmed path
Tool nose radius center path
Start point

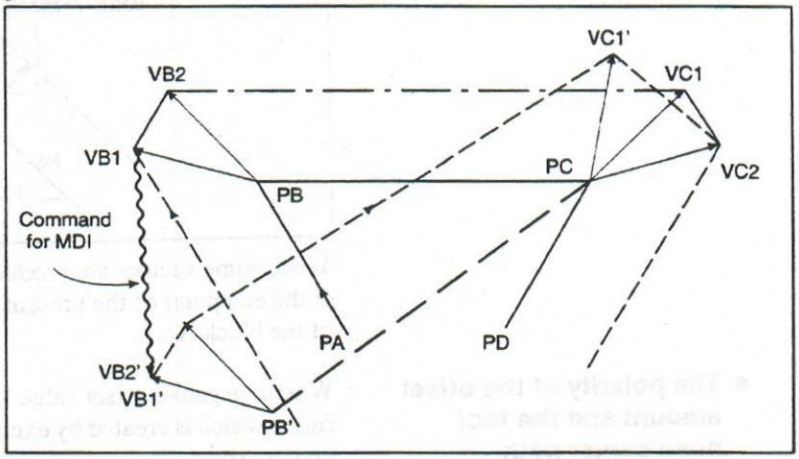
In outer chamfering with an offset, a limit is imposed on the programmed path. The path during chamfering coincides with the intersection points P_1 or P_2 without chamfering, therefore, outer chamfering is limited. In the figure above, the end point of the tool center path with chamfering coincides with the intersection point (P2) of the next block without chamfering. If the chamfering value is more than the limit value specified, alarm No.52 or 55 will be displayed.

14.3.8 Input Command from MDI

Tool nose radius compensation is not performed for commands input from the MDI.

However, when automatic operation using the CNC tape composed of absolute commands is temporarily stopped by the single block function, MDI operation is performed, then automatic operation starts again, the tool path is as follows :

In this case, the vectors at the start position of the next block are translated and the other vectors are produced by the next two blocks. Therefore, from next block but one, tool nose radius compensation is accurately performed.



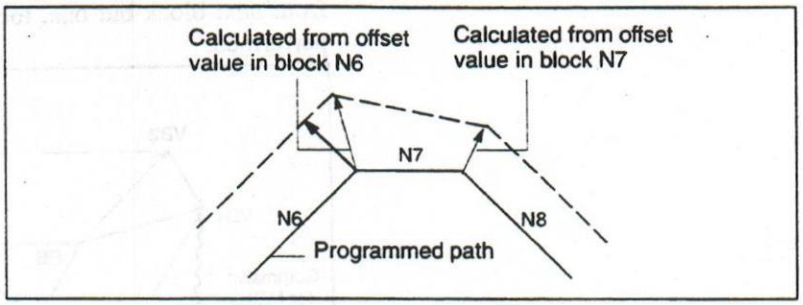
When position P_A, P_B, and P_C are programmed in an absolute command, tool is stopped by the single block function after executing the block from P_A to P_B and the tool is moved by MDI operation. Vectors V_{B1} and V_{B2} are translated to V_{B1}' and V_{B2}' and offset vectors are recalculated for the vectors V_{C1} and V_{C2} between block P_B–P_C and P_C–P_D.

However, since vector V_{B2} is not calculated again, compensation is accurately performed from position P_C.

14.3.9 General Precautions for Offset Operations

- **Changing the offset value**

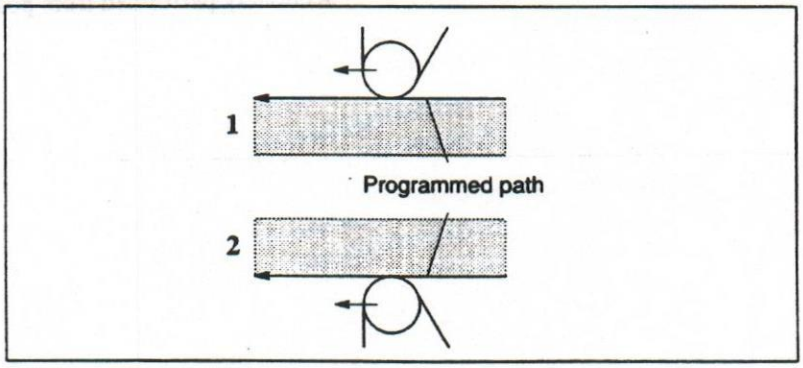
In general, the offset value is changed in offset cancel mode, or when changing tools. If the offset value is changed in offset mode, the vector at the end point of the block is calculated for the new offset value.



When some vectors are produced between blocks N6 and N7, the vector at the end point of the present blocks is calculated using the offset value of the block N6.

- **The polarity of the offset amount and the tool nose center path**

When a negative offset value is specified, the program is executed for the figure which is created by exchanging G41 for G42 or G42 for G41 in the process sheet.
A tool machining an inner profile will machine the occur profile, and tool machining the outer profile will machine the inner profile.
An example is shown below. In general, CNC machining is programmed assuming a positive offset value. When a program specifies a tool path as shown in **1**, the tool will move as shown in **2** if a negative offset is specified. The tool in **2** will move as shown in **1** when the sign of the offset value is reserved.



Notes

When the sign of the offset value is reversed, the offset vector of the tool nose is reversed but the imaginary tool nose direction does not change.
Therefore, do not reverse the sign of the offset value when starting the machining meeting the imaginary tool nose to the start point.

14.4 TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10)

Tool compensation values include tool geometry compensation values and tool wear compensation (Fig. 14.4 (a)).
Tool compensation can be specified without differentiating compensation for tool geometry from that for tool wear.

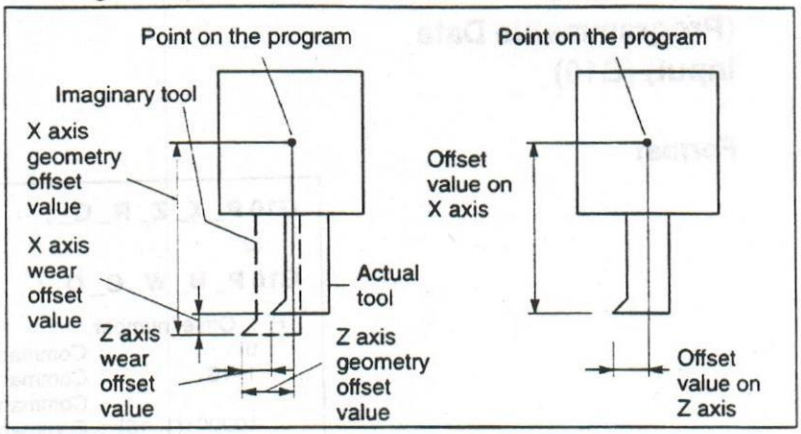

Point on the program
Imaginary tool
X axis geometry offset value
X axis wear offset value
Z axis wear offset value
Actual tool
Z axis geometry offset value
Point on the program
Offset value on X axis
Offset value on Z axis

Fig. 14.4(a) Difference the tool geometry offset from tool wear offset

Fig. 14.4(b) Not difference the tool geometry offset from tool wear offset

Tool compensation values can be entered into CNC memory from the CRT/MDI panel (see section III–11.4.1) or from a program.
A tool compensation value is selected from the CNC memory when the corresponding code is specified after address T in a program.
The value is used for tool offset or tool nose radius compensation.
See subsec. 14.1.2 for details.

14.4.1 Tool Compensation and Number of Tool Compensation

- **Valid range of tool compensation values**

Table 14.4.1 shows the valid input range of tool compensation values.

Table 14.4.1 Valid range of tool compensation values

Increment system	Tool compensation value	
	Metric input (mm)	Inch input (inch)
IS–B	–999.999 to +999.999 mm	–99.9999 to +99.9999 inch

The maximum tool wear compensation can be changed by setting parameter No.5013.

- **Number of tool compensation**

The memory can hold 16 tool compensation values.

14.4.2 Changing of Tool Offset Value (Programmable Data Input) (G10)

Offset values can be input by a program using the following command :

Format

```
G10 P_ X_ Z_ R_ Q_ ;
   or
G10 P_ U_ W_ C_ Q_ ;
```

P : Offset number
- 0 : Command of work coordinate system shift value
- 1–16 : Command of tool wear offset value
  Command value is offset number
- 10000+(1–16) : Command of tool geometry offset value
  (1–16) : Offset number

X : Offset value on X axis (absolute)
Z : Offset value on Z axis (absolute)
U : Offset value on X axis (incremental)
W : Offset value on Z axis (incremental)
R : Tool nose radius offset value (absolute)
R : Tool nose radius offset value (incremental)
Q : Imaginary tool nose number

In an absolute command, the values specified in addresses X, Z, and R are set as the offset value corresponding to the offset number specified by address P. In an incremental command, the value specified in addresses U, W, and C is added to the current offset value corresponding to the offset number.

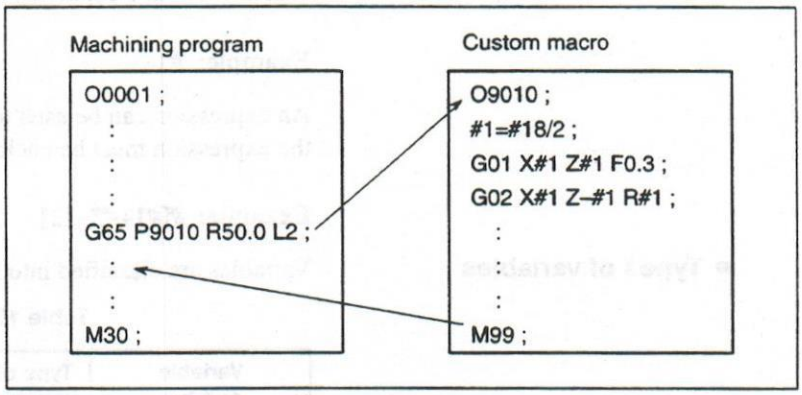
Notes

Notes

1. Addresses X, Z, U, and W can be specified in the same block.
2. Use of this command in a program allows the tool to advance little by little. This command can also be used input offset values one at a time from a program by specifying this command successively instead of inputting these values one at a time from the MDI unit.

15 CUSTOM MACRO

Although subprograms are useful for repeating the same operation, the custom macro function also allows use of variables, arithmetic and logic operations, and conditional branches for easy development of general programs such as pocketing and user–defined canned cycles.A machining program can call a custom macro with a simple command, just like a subprogram.



15.1 VARIABLES

An ordinary machining program specifies a G code and the travel distance directly with a numeric value; examples are G100 and X100.0.
With a custom macro, numeric values can be specified directly or using a variable number. When a variable number is used, the variable value can be changed by a program or using operations on the MDI panel.

```
#1=#2+100 ;
G01 X#1 F0.3 ;
```

Explanation

- **Variable representation**

When specifying a variable, specify a number sign (#) followed by a variable number. Personal computers allow a name to be assigned to a variable, but this capability is not available for custom macros.

Example: #1

An expression can be used to specify a variable number. In such a case, the expression must be enclosed in brackets.

Example: #[#1+#2-12]

- **Types of variables**

Variables are classified into four types by variable number.

Table 15.1 Types of variables

Variable number	Type of variable	Function
#0	Always null	This variable is always null. No value can be assigned to this variable.
#1 – #33	Local variables	Local variables can only be used within a macro to hold data such as the results of operations. When the power is turned off, local variables are initialized to null. When a macro is called, arguments are assigned to local variables.
#100 – #149 #500 – #531	Common variables	Common variables can be shared among different macro programs. When the power is turned off, variables #100 to #149 are initialized to null. Variables #500 to #531 hold data even when the power is turned off.
#1000 –	System variables	System variables are used to read and write a variety of NC data items such as the current position and tool compensation values.

- **Range of variable values**

Local and common variables can have value 0 or a value in the following ranges :

-10^{47} to -10^{-29}

0

$+10^{-29}$ to $+10^{47}$

If the result of calculation turns out to be invalid, an alarm No. 111 is issued.

- **Omission of the decimal point**

When a variable value is defined in a program, the decimal point can be omitted.

Example:

When #1=123; is defined, the actual value of variable #1 is 123.000.

- **Referencing variables**

To reference the value of a variable in a program, specify a word address followed by the variable number. When an expression is used to specify a variable, enclose the expression in brackets.

Example: G01X[#1+#2]F#3;

A referenced variable value is automatically rounded according to the least input increment of the address.

Example:

When G00X#1; is executed on a 1/1000–mm CNC with 12.3456 assigned to variable #1, the actual command is interpreted as G00X12.346;.

To reverse the sign of a referenced variable value, prefix a minus sign (–) to #.

Example: G00X–#1;

When an undefined variable is referenced, the variable is ignored up to an address word.

Example:

When the value of variable #1 is 0, and the value of variable #2 is null, execution of G00X#1Z#2; results in G00X0;.

- **Undefined variable**

When the value of a variable is not defined, such a variable is referred to as a "null" variable. Variable #0 is always a null variable. It cannot be written to, but it can be read.

(a) Quotation

When an undefined variable is quotated, the address itself is also ignored.

When #1 = < vacant >	When #1 = 0
G00 X100 Z#1 ↓ G00 X100	G00 X100 Z#1 ↓ G00 X100 Z0

(b) Operation

< vacant > is the same as 0 except when replaced by < vacant>

When #1 = < vacant >	When #1 = 0
#2 = #1 ↓ #2 = < vacant >	#2 = #1 ↓ #2 = 0
#2 = #1*5 ↓ #2 = 0	#2 = #1*5 ↓ #2 = 0
#2 = #1+#1 ↓ #2 = 0	#2 = #1 + #1 ↓ #2 = 0

(c) Conditional expressions

< vacant > differs from 0 only for EQ and NE.

When #1 = < vacant >	When #1 = 0
#1 EQ #0 ↓ Established	#1 EQ #0 ↓ Not established
#1 NE 0 ↓ Established	#1 NE 0 ↓ Not established
#1 GE #0 ↓ Established	#1 GE #0 ↓ Established
#1 GT 0 ↓ Not established	#1 GT 0 ↓ Not established

- **Displaying variable values**

Procedure for displaying variable values

1 Press the OFFSET SETTING key to display the tool compensation screen.

2 Press the continuous menu key ▷.

3 Press the soft key **[MACRO]** to display the macro variable screen.

4 Enter a variable number, then press soft key **[NO.SRH]**.
The cursor moves to the position of the entered number.

```
VARIABLE                                    O1234 N12345
  NO.        DATA          NO.              DATA
  100        123.456       108
  101          0.000       109
  102                      110
  103                      111
  104                      112
  105                      113
  106                      114
  107                      115

ACTUAL POSITION (RELATIVE)
    U          0.000          W          0.000


MEM **** *** ***           18:42:15

[ MACRO ] [      ] [ OPR ] [      ] [ (OPRT) ]
```

·When the value of a variable is blank, the variable is null.
·The mark ******** indicates an overflow (when the absolute value of a variable is greater than 99999999) or an underflow (when the absolute value of a variable is less than 0.0000001).

Limitations

Program numbers, and sequence numbers, cannot be referenced using variables.

Example:

Variables cannot be used in the following ways:
O#1;
N#3Z200.0;

15.2 SYSTEM VARIABLES

System variables can be used to read and write internal NC data such as tool compensation values and current position data. Note, however, that some system variables can only be read. System variables are essential for automation and general–purpose program development.

Explanations

- **Interface signals**

Signals can be exchanged between the programmable machine controller (PMC) and custom macros.

Table 15.2(a) System variables for interface signals

Variable number	Function
#1000–#1015 #1032	A 16–bit signal can be sent from the PMC to a custom macro. Variables #1000 to #1015 are used to read a signal bit by bit. Variable #1032 is used to read all 16 bits of a signal at one time.
#1100–#1115 #1132	A 16–bit signal can be sent from a custom macro to the PMC. Variables #1100 to #1115 are used to write a signal bit by bit. Variable #1132 is used to write all 16 bits of a signal at one time.
#1133	Variable #1133 is used to write all 32 bits of a signal at one time from a custom macro to the PMC. Note, that values from –99999999 to +99999999 can be used for #1133.

For detailed information, refer to the connection manual (B–62173E).

- **Tool compensation values**

When the system does not differentiate tool geometry compensation from tool wear compensation, use variable numbers for wear compensation.

Table 15.2(b) System variables for tool compensation memory

Compensation number	X axis compensation value		Z axis compensation value		Tool nose radius compensation value		Imaginary tool nose position
	Wear	Geometry	Wear	Geometry	Wear	Geometry	
1	#2001	#2701	#2101	#2801	#2201	#2901	#2301
:	:	:	:	:	:	:	:
16	:#2016	#2716	:#2116	#2816	:#2216	:#2916	#2316

Table 15.2(c) System variables for tool compensation memory

Compensation number	X axis compensation value		Z axis compensation value		Tool nose radius compensation value		Imaginary tool nose position
	Wear	Geometry	Wear	Geometry	Wear	Geometry	
1	#10001	#15001	#11001	#16001	#12001	#17001	#13001
:	:	:	:	:	:	:	:
:	:	:	:	:	:	:	:
16	#10016	#15016	#11016	#16016	#12016	#17016	#13016

- **Macro alarms**

Table 15.2(c) System variable for macro alarms

Variable number	Function
#3000	When a value from 0 to 200 is assigned to variable #3000, the NC stops with an alarm. After an expression, an alarm message not longer than 26 characters can be described. The CRT screen displays alarm numbers by adding 3000 to the value in variable #3000 along with an alarm message.

Example:

#3000=1(TOOL NOT FOUND);

→ The alarm screen displays "3001 TOOL NOT FOUND."

- **Time information**

Time information can be read and written.

Table 15.2(d) System variables for time information

Variable number	Function
#3001	This variable functions as a timer that counts in 1–millisecond increments at all times. When the power is turned on, the value of this variable is reset to 0. When 65535 milliseconds is reached, the value of this timer returns to 0.
#3002	This variable functions as a timer that counts in 1–hour increments when the cycle start lamp is on. This timer preserves its value even when the power is turned off. When 1145324.612 hours is reached, the value of this timer returns to 0.
#3011	This variable can be used to read the current date (year/month/day). Year/month/day information is converted to an apparent decimal number. For example, March 28, 1993 is represented as 19930328.
#3012	This variable can be used to read the current time (hours/minutes/seconds). Hours/minutes/seconds information is converted to an apparent decimal number. For example, 34 minutes and 56 seconds after 3 p.m. is represented as 153456.

- **Automatic operation control**

The control state of automatic operation can be changed.

Table 15.2(e) System variable (#3003) for automatic operation control

#3003	Single block	Completion of an auxiliary function
0	Enabled	To be awaited
1	Disabled	To be awaited
2	Enabled	Not to be awaited
3	Disabled	Not to be awaited

- When the power is turned on, the value of this variable is 0.
- When single block stop is disabled, single block stop operation is not performed even if the single block switch is set to ON.

- When a wait for the completion of auxiliary functions (M, S, and T functions) is not specified, program execution proceeds to the next block before completion of auxiliary functions. Also, distribution completion signal DEN is not output.

Table 15.2(f) System variable (#3004) for automatic operation control

#3004	Feed hold	Feedrate Override	Exact stop
0	Enabled	Enabled	Enabled
1	Disabled	Enabled	Enabled
2	Enabled	Disabled	Enabled
3	Disabled	Disabled	Enabled
4	Enabled	Enabled	Disabled
5	Disabled	Enabled	Disabled
6	Enabled	Disabled	Disabled
7	Disabled	Disabled	Disabled

- When the power is turned on, the value of this variable is 0.
- When feed hold is disabled:
  (1) When the feed hold button is held down, the machine stops in the single block stop mode. However, single block stop operation is not performed when the single block mode is disabled with variable #3003.
  (2) When the feed hold button is pressed then released, the feed hold lamp comes on, but the machine does not stop; program execution continues and the machine stops at the first block where feed hold is enabled.
- When feedrate override is disabled, an override of 100% is always applied regardless of the setting of the feedrate override switch on the machine operator's panel.
- When exact stop check is disabled, no exact stop check (position check) is made even in blocks including those which do not perform cutting.

• Settings

Settings can be read and written. Binary values are converted to decimals.

#3005

	#15	#14	#13	#12	#11	#10	#9	#8
Setting								

	#7	#6	#5	#4	#3	#2	#1	#0
Setting			SEQ			INI	ISO	TVC

#5 (SEQ) : Whether to automatically insert sequence numbers
#2 (INI) : Millimeter input or inch input
#1 (ISO): Whether to use EIA or ISO as the output code
#0 (TVC) : Whether to make a TV check

- **Mirror image**

The mirror–image status for each axis set using an external switch or setting operation can be read through the output signal (mirror–image check signal). The mirror–image status present at that time can be checked. (See Section 4.5 in III.)
The value obtained in binary is converted into decimal notation.

#3007

	#7	#6	#5	#4	#3	#2	#1	#0
Setting							2th axis	1th axis

For each bit, 0 (mirror–image function is disabled) or 1 (mirror–image function is enabled) is indicated.

Example : If #3007 is 3, the mirror–image function is enabled for the first and second axes.

- When the mirror–image function is set for a certain axis by both the mirror–image signal and setting, the signal value and setting value are ORed and then output.
- When mirror–image signals for axes other than the controlled axes are turned on, they are still read into system variable #3007.
- System variable #3007 is a write–protected system variable. If an attempt is made to write data in the variable, P/S 116 alarm "WRITE PROTECTED VARIABLE" is issued.

- **Number of machined parts**

The number (target number) of parts required and the number (completion number) of machined parts can be read and written.

Table 15.2(g) System variables for the number of parts required and the number of machined parts

Variable number	Function
#3901	Number of machined parts (completion number)
#3902	Number of required parts (target number)

Notes
Do not substitute a negative value.

• Modal information

Modal information specified in blocks up to the immediately preceding block can be read.

Table 15.2(h) System variables for modal information

Variable number	Function	
#4001	G00, G01, G02, G03, G32	(Group 01)
#4002	G96, G97	(Group 02)
#4003		(Group 03)
#4004		(Group 04)
#4005	G98, G99	(Group 05)
#4006	G20, G21	(Group 06)
#4007	G40, G41, G42	(Group 07)
#4008		(Group 08)
#4009		(Group 09)
#4010		(Group 10)
#4011		(Group 11)
#4012	G66, G67	(Group 12)
#4014		(Group 14)
#4015		(Group 15)
#4016		(Group 16)
:	:	:
#4022		(Group 22)
#4109	F code	
#4113	M code	
#4114	Sequence number	
#4115	Program number	
#4119	S code	
#4120	T code	

Example:

When #1=#4005; is executed, the resulting value in #1 is 98 or 99.

• Current position

Position information cannot be written but can be read.

Table 15.2(i) System variables for position information

Variable number	Position information	Coordinate system	Tool compensation value	Read operation during movement
#5001–#5002	Block end point	Workpiece coordinate system	Not included	Enabled
#5021–#5022	Current position	Machine coordinate system	Included	Disabled
#5041–#5042	Current position	Workpiece coordinate system		
#5061–#5062	Skip signal position			Enabled
#5081, #5082	Tool offset value			Disabled
#5101–#5102	Deviated servo position			

- The first digit (from 1 to 2) represents an axis number.
- The tool offset value currently used for execution rather than the immediately preceding tool offset value is held in variables #5081 to 5088.
- The tool position where the skip signal is turned on in a G31 (skip function) block is held in variables #5061 to #5068. When the skip signal is not turned on in a G31 block, the end point of the specified block is held in these variables.
- When read during movement is "disabled," this means that expected values cannot be read due to the buffering (preread) function.

15.3 ARITHMETIC AND LOGIC OPERATION

The operations listed in Table 15.3(a) can be performed on variables. The expression to the right of the operator can contain constants and/or variables combined by a function or operator. Variables #j and #K in an expression can be replaced with a constant. Variables on the left can also be replaced with an expression.

Table 15.3(a) Arithmetic and logic operation

Function	Format	Remarks
Definition	#i=#j	
Sum Difference Product Quotient	#i=#j+#k; #i=#j-#k; #i=#j*#k; #i=#j/#k;	
Sine Cosine Tangent Arctangent	#i=SIN[#j]; #i=COS[#j]; #i=TAN[#j]; #i=ATAN[#j]/[#k] ;	An angle is specified in degrees. 90 degrees and 30 minutes is represented as 90.5 degrees.
Square root Absolute value Rounding off Rounding down Rounding up	#i=SQRT[#j]; #i=ABS[#j]; #i=ROUND[#j]; #i=FIX[#j]; #i=FUP[#j];	
OR XOR AND	#i=#j OR #k; #i=#j XOR #k; #i=#j AND #k;	A logical operation is performed on binary numbers bit by bit.
Conversion from BCD to BIN Conversion from BIN to BCD	#i=BIN[#j]; #i=BCD[#j];	Used for signal exchange to and from the PMC

Explanations

- **Angle units**

The units of angles used with the SIN, COS, TAN, and ATAN functions are degrees. For example, 90 degrees and 30 minutes is represented as 90.5 degrees.

- **ATAN function**

After the ATAN function, specify the lengths of two sides separated by a slash. A result is found where $0 \leqq \text{result} < 360$.

Example :
When #1=ATAN[1]/[-1], the value of #1 is 135.0

- **ROUND function**

When the ROUND function is included in an arithmetic or logic operation command, IF statement, or WHILE statement, the ROUND function rounds off at the first decimal place.

Example:
When #1=ROUND[#2]; is executed where #2 holds 1.2345, the value of variable #1 is 1.0.

·When the ROUND function is used in NC statement addresses, the ROUND function rounds off the specified value according to the least input increment of the address.

Example:

Creation of a drilling program that cuts according to the values of variables #1 and #2, then returns to the original position
Suppose that the increment system is 1/1000 mm, variable #1 holds 1.2345, and variable #2 holds 2.3456. Then,
G00 G91 X–#1; Moves 1.235 mm.
G01 X–#2 F300; Moves 2.346 mm.
G00 X[#1+#2];
Since 1.2345 + 2.3456 = 3.5801, the travel distance is 3.580, which does not return the tool to the original position.
This difference comes from whether addition is performed before or after rounding off. G00X–[ROUND[#1]+ROUND[#2]] must be specified to return the tool to the original position.

• Rounding up and down to an integer

With NC, when the absolute value of the integer produced by an operation on a number is greater than the absolute value of the original number, such an operation is referred to as rounding up to an integer. Conversely, when the absolute value of the integer produced by an operation on a number is less than the absolute value of the original number, such an operation is referred to as rounding down to an integer. Be particularly careful when handling negative numbers.

Example:

Suppose that #1=1.2 and #2=–1.2.
When #3=FUP[#1] is executed, 2.0 is assigned to #3.
When #3=FIX[#1] is executed, 1.0 is assigned to #3.
When #3=FUP[#2] is executed, –2.0 is assigned to #3.
When #3=FIX[#2] is executed, –1.0 is assigned to #3.

• Abbreviations of arithmetic and logic operation commands

When a function is specified in a program, the first two characters of the function name can be used to specify the function.

Example:

ROUND → RO
FIX → FI

• Priority of operations

(1) Functions
(2) Operations such as multiplication and division (*, /, AND)
(3) Operations such as addition and subtraction (+, –, OR, XOR)

Example) #1=#2+#3*SIN[#4];

(1)
(2)
(3)

(1), (2), and (3) indicate the order of operations.

- **Bracket nesting**

Brackets are used to change the order of operations. Brackets can be used to a depth of five levels including the brackets used to enclose a function. When a depth of five levels is exceeded, alarm No. 118 occurs.

Example) #1=SIN [[[#2+#3] *#4 +#5] *#6] ;

(1)
(2)
(3)
(4)
(5)

(1) to (5) indicate the order of operations.

Limitations

- **Brackets**

Brackets ([,]) are used to enclose an expression. Note that parentheses are used for comments.

- **Operation error**

Errors may occur when operations are performed.

Table 15.3(b) Errors involved in operations

Operation	Average error	Maximum error	Type of error
a = b*c	1.55×10^{-10}	4.66×10^{-10}	$a = \sqrt{b}$
a = b / c	4.66×10^{-10}	1.88×10^{-9}	
	1.24×10^{-9}	3.73×10^{-9}	
a = b + c a = b − c	2.33×10^{-10}	5.32×10^{-10}	$\text{Min}\left\lvert\frac{\varepsilon}{b}\right\rvert,\left\lvert\frac{\varepsilon}{c}\right\rvert$ (*2)
a = SIN [b] a = COS [b]	5.0×10^{-9}	1.0×10^{-8}	Absolute error(*3) $\lvert\varepsilon\rvert$ degrees
a = ATAN [b] / [c] (*4)	1.8×10^{-6}	3.6×10^{-6}	

Notes
1. The relative error depends on the result of the operation.
2. Smaller of the two types of errors is used.
3. The absolute error is constant, regardless of the result of the operation.
4. Function TAN performs SIN/COS.

- The precision of variable values is about 8 decimal digits. When very large numbers are handled in an addition or subtraction, the expected results may not be obtained.

  Example:

  When an attempt is made to assign the following values to variables #1 and #2:

  #1=9876543210123.456

  #2=9876543277777.777

  the values of the variables become:

  #1=9876543200000.000

  #2=9876543300000.000

  In this case, when #3=#2–#1; is calculated, #3=100000.000 results. (The actual result of this calculation is slightly different because it is performed in binary.)

- Also be aware of errors that can result from conditional expressions using EQ, NE, GE, GT, LE, and LT.

  Example:

  IF[#1 EQ #2] is effected by errors in both #1 and #2, possibly resulting in an incorrect decision.

  Therefore, instead find the difference between the two variables with IF[ABS[#1–#2]LT0.001].

  Then, assume that the values of the two variables are equal when the difference does not exceed an allowable limit (0.001 in this case).

- Also, be careful when rounding down a value.

  Example:

  When #2=#1*1000; is calculated where #1=0.002;, the resulting value of variable #2 is not exactly 2 but 1.99999997.

  Here, when #3=FIX[#2]; is specified, the resulting value of variable #1 is not 2.0 but 1.0. In this case, round down the value after correcting the error so that the result is greater than the expected number, or round it off as follows:

  #3=FIX[#2+0.001]

  #3=ROUND[#2]

- **Divisor**

When a divisor of zero is specified in a division or TAN[90], alarm No. 112 occurs.

15.4 MACRO STATEMENTS AND NC STATEMENTS

The following blocks are referred to as macro statements:

- **Blocks containing an arithmetic or logic operation (=)**
- **Blocks containing a control statement (such as GOTO, DO, END)**
- **Blocks containing a macro call command (such as macro calls by G65, G66, G67, or other G codes, or by M codes)**

Any block other than a macro statement is referred to as an NC statement.

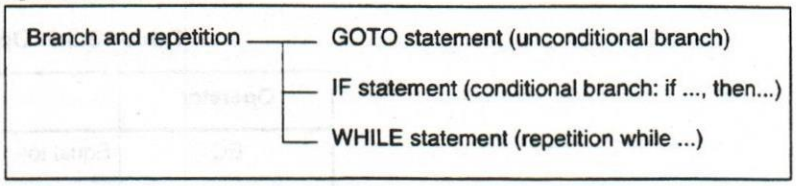
Explanations

- **Differences from NC statements**

  - Even when single block mode is on, the machine does not stop. Note, however, that the machine stops in the single block mode when bit 5 of parameter 6000 is 1.
  - Macro blocks are not regarded as blocks that involve no movement in the tool nose radius compensation mode (see Section 15.7).

15.5 BRANCH AND REPETITION

In a program, the flow of control can be changed using the GOTO statement and IF statement. Three types of branch and repetition operations are used:


Branch and repetition
GOTO statement (unconditional branch)
IF statement (conditional branch: if ..., then...)
WHILE statement (repetition while ...)

15.5.1 Unconditional Branch (GOTO Statement)

A branch to sequence number n occurs. When a sequence number outside of the range 1 to 99999 is specified, alarm No. 128 occurs. A sequence number can also be specified using an expression.

```
GOTO n ;    n: Sequence number (1 to 99999)
```

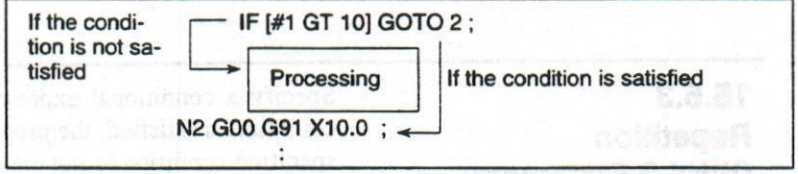
Example:

```
GOTO1;
GOTO#10;
```

15.5.2 Conditional Branch (IF Statement)

Specify a conditional expression after IF. If the specified conditional expression is satisfied, a branch to sequence number n occurs. If the specified condition is not satisfied, the next block is executed.

If the value of variable #1 is greater than 10, a branch to sequence number N2 occurs.



Explanations

- **Conditional expression**

A conditional expression must include an operator inserted between two variables or between a variable and constant, and must be enclosed in brackets ([,]). An expression can be used instead of a variable.

• Operators

Operators each consist of two letters and are used to compare two values to determine whether they are equal or one value is smaller or greater than the other value. Note that the inequality sign cannot be used.

Table 15.5.2 Operators

Operator	Meaning
EQ	Equal to(=)
NE	Not equal to(≠)
GT	Greater than(>)
GE	Greater than or equal to(≧)
LT	Less than(<)
LE	Less than or equal to(≦)

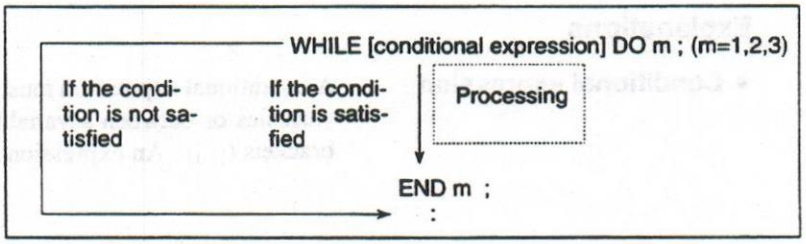
Sample program

The sample program below finds the total of numbers 1 to 10.

```
O9500;
 #1=0;Initial value of the variable to hold the sum
 #2=1;Initial value of the variable as an addend
N1 IF[#2 GT 10] GOTO 2;  Branch to N2 when the addend is greater than
10
 #1=#1+#2;  Calculation to find the sum
 #2=#2+1; Next addend
 GOTO 1;  Branch to N1
N2 M30;End of program
```

15.5.3 Repetition (WHILE Statement)

Specify a conditional expression after WHILE. While the specified condition is satisfied, the program from DO to END is executed. If the specified condition is not satisfied, program execution proceeds to the block after END.

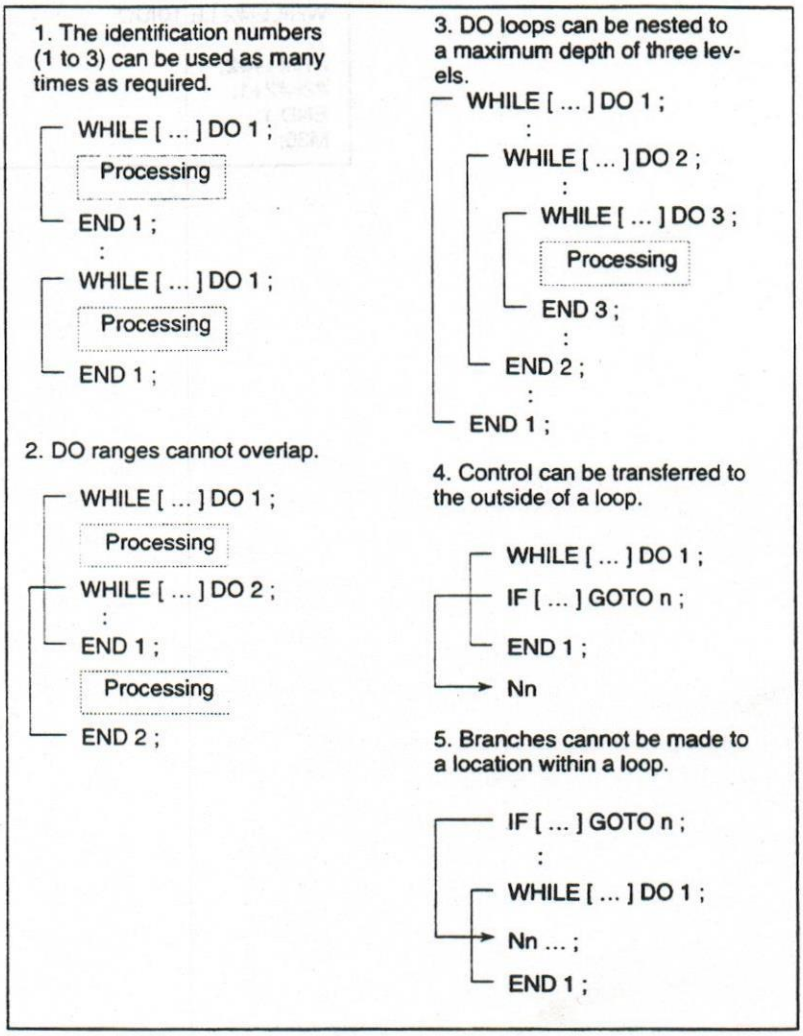

WHILE [conditional expression] DO m ; (m=1,2,3)
If the condition is not satisfied
If the condition is satisfied
Processing
END m ;

Explanations

While the specified condition is satisfied, the program from DO to END after WHILE is executed. If the specified condition is not satisfied, program execution proceeds to the block after END. The same format as for the IF statement applies. A number after DO and a number after END are identification numbers for specifying the range of execution. The numbers 1, 2, and 3 can be used. When a number other than 1, 2, and 3 is used, alarm No. 126 occurs.

- **Nesting**

The identification numbers (1 to 3) in a DO–END loop can be used as many times as desired. Note, however, when a program includes crossing repetition loops (overlapped DO ranges), alarm No. 124 occurs.


1. The identification numbers (1 to 3) can be used as many times as required.
WHILE [...] DO 1 ;
Processing
END 1 ;
:
WHILE [...] DO 1 ;
Processing
END 1 ;
2. DO ranges cannot overlap.
WHILE [...] DO 1 ;
Processing
WHILE [...] DO 2 ;
:
END 1 ;
Processing
END 2 ;
3. DO loops can be nested to a maximum depth of three levels.
WHILE [...] DO 1 ;
:
WHILE [...] DO 2 ;
:
WHILE [...] DO 3 ;
Processing
END 3 ;
:
END 2 ;
:
END 1 ;
4. Control can be transferred to the outside of a loop.
WHILE [...] DO 1 ;
IF [...] GOTO n ;
END 1 ;
Nn
5. Branches cannot be made to a location within a loop.
IF [...] GOTO n ;
:
WHILE [...] DO 1 ;
Nn ... ;
END 1 ;

Limitations

- **Infinite loops**

When DO m is specified without specifying the WHILE statement, an infinite loop ranging from DO to END is produced.

- **Processing time**

When a branch to the sequence number specified in a GOTO statement occurs, the sequence number is searched for. For this reason, processing in the reverse direction takes a longer time than processing in the forward direction. Using the WHILE statement for repetition reduces processing time.

- **Undefined variable**

In a conditional expression that uses EQ or NE, a null value and zero have different effects. In other types of conditional expressions, a null value is regarded as zero.

Sample program

The sample program below finds the total of numbers 1 to 10.

```
O0001;
#1=0;
#2=1;
WHILE[#2 LE 10]DO
1;
#1=#1+#2;
#2=#2+1;
END 1;
M30;
```

15.6 MACRO CALL

A macro program can be called using the following methods:

- Macro call
  - Simple call ((G65)
  - modal call (G66, G67)
  - Macro call with G code
  - Macro call with M code
  - Subprogram call with M code
  - Subprogram call with T code

Restrictions

- **Differences between macro calls and subprogram calls**

Macro call (G65) differs from subprogram call (M98) as described below.

- With G65, an argument (data passed to a macro) can be specified. M98 does not have this capability.
- When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the subprogram is called after the command is executed. On the other hand, G65 unconditionally calls a macro.
- When an M98 block contains another NC command (for example, G01 X100.0 M98Pp), the machine stops in the single block mode. On the other hand, G65 does not stops the machine.
- With G65, the level of local variables changes. With M98, the level of local variables does not change.

15.6.1 Simple Call (G65)

When G65 is specified, the custom macro specified at address P is called. Data (argument) can be passed to the custom macro program.

```
G65 P p L  ℓ <argument-specification>
;
  P : Number of the program to call
  ℓ : Repetition count (1 by default)
  Argument : Data passed to the macro
```

```
O0001 ;                    O9100 ;
   :                       #3=#1+#2 ;
G65 P9010 L2 A1.0 B2.0 ;   IF [#3 GT 360] GOTO 9 ;
   :                       G00 X#3 ;
M30 ;                      N9 M99 ;
```

Explanations

- **Call**

- After G65, specify at address P the program number of the custom macro to call.
- When a number of repetitions is required, specify a number from 1 to 9999 after address L. When L is omitted, 1 is assumed.
- By using argument specification, values are assigned to corresponding local variables.

- **Argument specification**

Two types of argument specification are available. Argument specification I uses letters other than G, L, O, N, and P once each. Argument specification II uses A, B, and C once each and also uses I, J, and K up to ten times. The type of argument specification is determined automatically according to the letters used.

Argument specification I

Address	Variable number
A	#1
B	#2
C	#3
D	#7
E	#8
F	#9
H	#11

Address	Variable number
I	#4
J	#5
K	#6
M	#13
Q	#17
R	#18
S	#19

Address	Variable number
T	#20
U	#21
V	#22
W	#23
X	#24
Y	#25
Z	#26

- Addresses G, L, N, O, and P cannot be used in arguments.
- Addresses that need not be specified can be omitted. Local variables corresponding to an omitted address are set to null.

Argument specification II

Argument specification II uses A, B, and C once each and uses I, J, and K up to ten times. Argument specification II is used to pass values such as three–dimensional coordinates as arguments.

Address	Variable number
A	#1
B	#2
C	#3
I_1	#4
J_1	#5
K_1	#6
I_2	#7
J_2	#8
K_2	#9
I_3	#10
J_3	#11

Address	Variable number
K_3	#12
I_4	#13
J_4	#14
K_4	#15
I_5	#16
J_5	#17
K_5	#18
I_6	#19
J_6	#20
K_6	#21
I_7	#22

Address	Variable number
J_7	#23
K_7	#24
I_8	#25
J_8	#26
K_8	#27
I_9	#28
J_9	#29
K_9	#30
I_{10}	#31
J_{10}	#32
K_{10}	#33

- Subscripts of I, J, and K for indicating the order of argument specification are not written in the actual program.

Restrictions

- **Format**

G65 must be specified before any argument.

- **Mixture of argument specifications I and II**

The NC internally identifies argument specification I and argument specification II. If a mixture of argument specification I and argument specification II is specified, the type of argument specification specified later takes precedence.

- **Position of the decimal point**

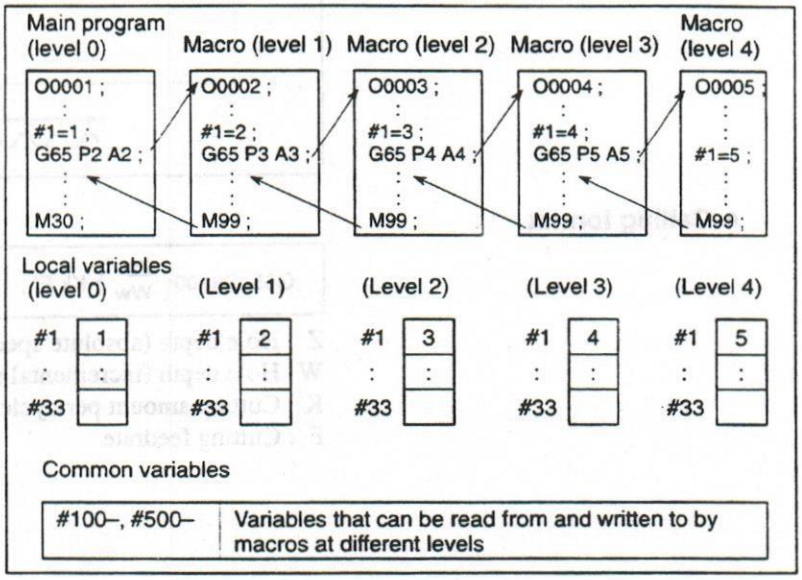
The units used for argument data passed without a decimal point correspond to the least input increment of each address. The value of an argument passed without a decimal point may vary according to the system configuration of the machine. It is good practice to use decimal points in macro call arguments to maintain program compatibility.

- **Call nesting**

Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).

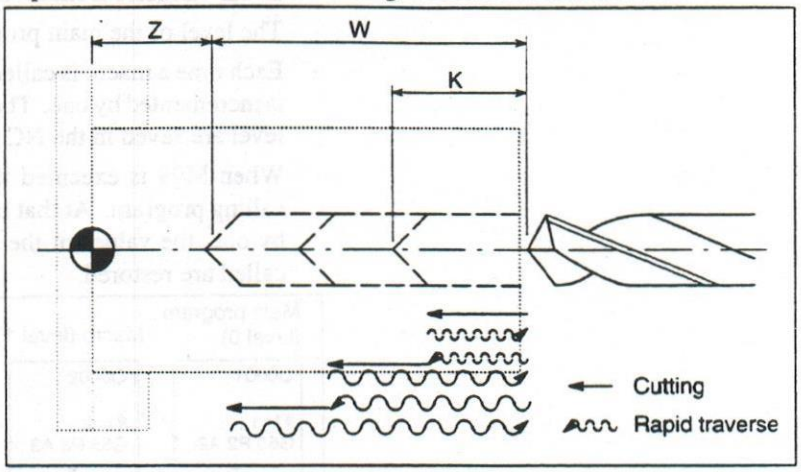
- **Local variable levels**

- Local variables from level 0 to 4 are provided for nesting.
- The level of the main program is 0.
- Each time a macro is called (with G65 or G66), the local variable level is incremented by one. The values of the local variables at the previous level are saved in the NC.
- When M99 is executed in a macro program, control returns to the calling program. At that time, the local variable level is decremented by one; the values of the local variables saved when the macro was called are restored.



Sample program (Drill cycle)

Move the tool beforehand along the X– and Z–axes to the position where a drilling cycle starts. Specify Z or W for the depth of a hole, K for the depth of a cut, and F for the cutting feedrate to drill the hole.



- **Calling format**

G65 P9100 { Zz / Ww } Kk Ff ;

Z : Hole depth (absolute specification)
W : Hole depth (incremental specification)
K : Cutting amount per cycle
F : Cutting feedrate

- **Program calling a macro program**

```
O0002;
G50 X100.0 Z200.0 ;
G00 X0 Z102.0 S1000 M03 ;
G65 P9100 Z50.0 K20.0 F0.3 ;
G00 X100.0 Z200.0 M05 ;
M30 ;
```

- **Macro program (called program)**

```
      O9100;
      #1=0 ; ..................  Clear the data for the depth of the current hole.
      #2=0 ; ..................  Clear the data for the depth of the preceding
                                 hole.
      IF [#23 NE #0] GOTO 1 ; ..  If incremental programming, specifies the
                                 jump to N1.
      IF [#26 EQ #0] GOTO 8 ; ..  If neither Z nor W is specified, an error
                                 occurs.
      #23=#5002-#26 ; .........  Calculates the depth of a hole.
N1    #1=#1+#6 ; ..............  Calculates the depth of the current hole.
      IF [#1 LE #23] GOTO 2 ; ..  Determines whether the hole to be cut is
                                 too deep.?
      #1=#23 ; ................  Clamps at the depth of the current hole.
N2    G00 W-#2 ; ..............  Moves the tool to the depth of the preceding
                                 hole at the cutting feedrate.
      G01 W- [#1-#2] F#9 ; ...  Drills the hole.
      G00 W#1 ; ...............  Moves the tool to the drilling start point.
      IF [#1 GE #23] GOTO 9 ; ..  Checks whether drilling is completed.
      #2=#1 ; .................  Stores the depth of the current hole.
      GOTO 1 ;
N9    M99 ;
N8    #3000=1 (NOT Z OR U COMMAND)
```

15.6.2 Modal Call (G66)

Once G66 is issued to specify a modal call a macro is called after a block specifying movement along axes is executed. This continues until G67 is issued to cancel a modal call.

```
G66 P p L  ℓ <argument-specification>
;
   P :  Number of the program to call
   ℓ :  Repetition count (1 by default)
   Argument :  Data passed to the macro
```

```
O0001 ;                          O9100 ;
   :                                :
G66 P9100 L2 A1.0 B2.0 ;         G00 Z-#1 ;
G00 G90 X100.0 ;                 G01 Z-#2 F0.3 ;
X125.0 ;                            :
X150.0 ;                            :
G67 ;                               :
   :                                :
M30 ;                            M99 ;
```

Explanations

- **Call**
  - After G66, specify at address P a program number subject to a modal call.
  - When a number of repetitions is required, a number from 1 to 9999 can be specified at address L.
  - As with a simple call (G65), data passed to a macro program is specified in arguments.
- **Cancellation**

  When a G67 code is specified, modal macro calls are no longer performed in subsequent blocks.
- **Call nesting**

  Calls can be nested to a depth of four levels including simple calls (G65) and modal calls (G66). This does not include subprogram calls (M98).
- **Modal call nesting**

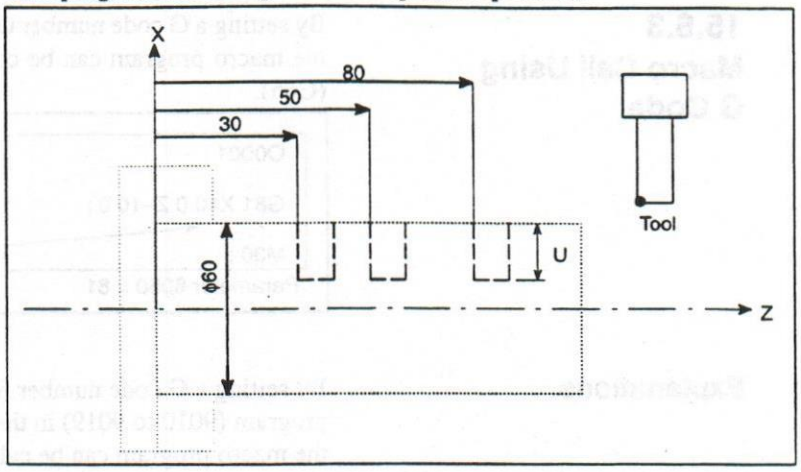
  Modal calls can be nested by specifying another G66 code during a modal call.

Restrictions

- In a G66 block, no macros can be called.
- G66 needs to be specified before any arguments.
- No macros can be called in a block which contains a code such as a miscellaneous function that does not involve movement along an axis.
- Local variables (arguments) can only be set in G66 blocks. Note that local variables are not set each time a modal call is performed.

Sample program

This program makes a groove at a specified position.



- **Calling format**

```
G66 P9110 Uu Ff ;
```

U: Groove depth (incremental specification)
F: Cutting feed of grooving

- **Program that calls a macro program**

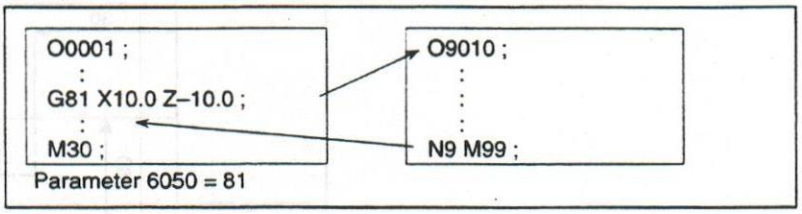
```
O0003 ;
G50 X100.0 Z200.0 ;
S1000 M03 ;
G66 P9110 U5.0 F0.5 ;
G00 X60.0 Z80.0 ;
Z50.0 ;
Z30.0 ;
G67 ;
G00 X00.0 Z200.0 M05 ;
M30;
```

- **Macro program (program called)**

```
O9110 ;
G01 U–#21 F#9 ; ..... Cuts the workpiece.
G00 U#21 ; ..... Retracts the tool.
M99 ;
```

15.6.3 Macro Call Using G Code

By setting a G code number used to call a macro program in a parameter, the macro program can be called in the same way as for a simple call (G65).



Explanations

By setting a G code number from 1 to 9999 used to call a custom macro program (9010 to 9019) in the corresponding parameter (6050 to 6059), the macro program can be called in the same way as with G65.
For example, when a parameter is set so that macro program O9010 can be called with G81, a user–specific cycle created using a custom macro can be called without modifying the machining program.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9010	6050
O9011	6051
O9012	6052
O9013	6053
O9014	6054
O9015	6055
O9016	6056
O9017	6057
O9018	6058
O9019	6059

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

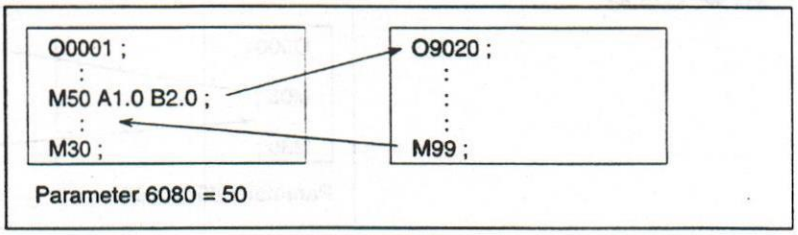
Restrictions

- **Nesting of calls using G codes**

In a program called with a G code, no macros can be called using a G code. A G code in such a program is treated as an ordinary G code. In a program called as a subprogram with an M or T code, no macros can be called using a G code. A G code in such a program is also treated as an ordinary G code.

15.6.4 Macro Call Using an M Code

By setting an M code number used to call a macro program in a parameter, the macro program can be called in the same way as with a simple call (G65).



Explanations

By setting an M code number from 1 to 99999999 used to call a custom macro program (9020 to 9029) in the corresponding parameter (6080 to 6089), the macro program can be called in the same way as with G65.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9020	6080
O9021	6081
O9022	6082
O9023	6083
O9024	6084
O9025	6085
O9026	6086
O9027	6087
O9028	6088
O9029	6089

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

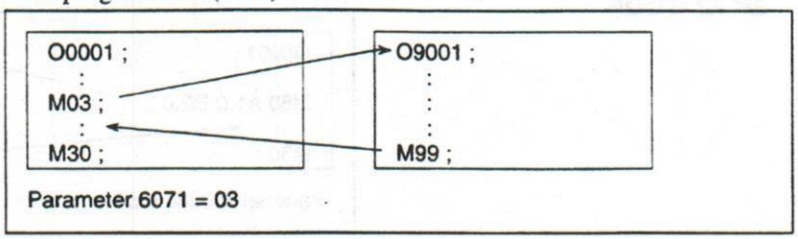
As with a simple call, two types of argument specification are available: Argument specification I and argument specification II. The type of argument specification is determined automatically according to the addresses used.

Restrictions

- An M code used to call a macro program must be specified at the start of a block.
- In a macro called with a G code or in a program called as a subprogram with an M or T code, no macros can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

15.6.5 Subprogram Call Using an M Code

By setting an M code number used to call a subprogram (macro program) in a parameter, the macro program can be called in the same way as with a subprogram call (M98).



Explanations

By setting an M code number from 1 to 99999999 used to call a subprogram in a parameter (6071 to 6076), the corresponding custom macro program (9001 to 9006) can be called in the same way as with M98.

- **Correspondence between parameter numbers and program numbers**

Program number	Parameter number
O9001	6071
O9002	6072
O9003	6073
O9004	6074
O9005	6075
O9006	6076
O9007	6077
O9008	6078
O9009	6079

- **Repetition**

As with a simple call, a number of repetitions from 1 to 9999 can be specified at address L.

- **Argument specification**

Argument specification is not allowed.

- **M code**

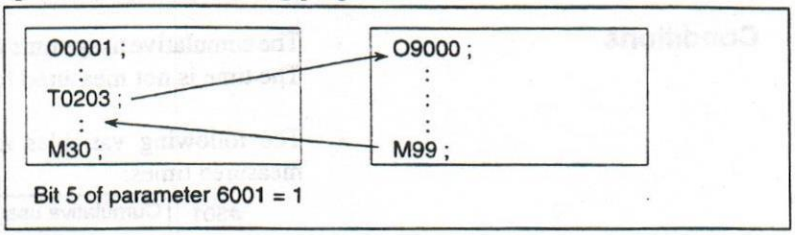
An M code in a macro program that has been called is treated as an ordinary M code.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using an M code. An M code in such a macro or program is treated as an ordinary M code.

15.6.6 Subprogram Calls Using a T Code

By enabling subprograms (macro program) to be called with a T code in a parameter, a macro program can be called each time the T code is specified in the machining program.



Explanations

- **Call**

By setting bit 5 of parameter 6001 to 1, the macro program O9000 can be called when a T code is specified in the machining program. A T code specified in a machining program is assigned to common variable #149.

Limitations

In a macro called with a G code or in a program called with an M or T code, no subprograms can be called using a T code. A T code in such a macro or program is treated as an ordinary T code.

15.6.7 Sample Program

By using the subprogram call function that uses M codes, the cumulative usage time of each tool is measured.

Conditions

- The cumulative usage time of each of tool numbers 1 to 5 is measured. The time is not measured for tools whose number is 6 or more.
- The following variables are used to store the tool numbers and measured times:

#501	Cumulative usage time of tool number 1
#502	Cumulative usage time of tool number 2
#503	Cumulative usage time of tool number 3
#504	Cumulative usage time of tool number 4
#505	Cumulative usage time of tool number 5

- Usage time starts being counted when the M03 command is specified and stops when M05 is specified. System variable #3002 is used to measure the time during which the cycle start lamp is on. The time during which the machine is stopped by feed hold and single block stop operation is not counted, but the time used to change tools and pallets is included.

Operation check

- **Parameter setting** — Set 3 in parameter 6071, and set 05 in parameter 6072.
- **Variable value setting** — Set 0 in variables #501 to #505.

• Program that calls a macro program

```
O0001;
T0100 M06;
M03;
 :
 :
M05; ..... Changes #501.
T0200 M06;
M03;
 :
 :
M05; ..... Changes #502.
T0300 M06;
M03;
 :
 :
M05; ..... Changes #503.
T0400 M06;
M03;
 :
 :
M05; ..... Changes #504.
T0500 M06;
M03;
 :
 :
M05; ..... Changes #505.
M30;
```

Macro program (program called)

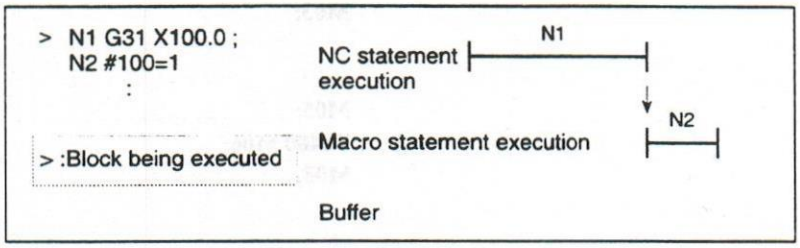
```
O9001(M03); ..... Macro to start counting
  M01;
  IF[FIX[#4120/100] EQ 0]GOTO 9; ..... No tool specified
  IF[FIX[#4120/100] GT 5]GOTO 9; ..... Out–of–range tool number
  #3002=0; ..... Clears the timer.
N9 M03; ..... Rotates the spindle in the forward direction.
  M99;

O9002(M05); ..... Macro to end counting
  M01;
  IF[FIX[#4120/100] EQ 0]GOTO 9; ..... No tool specified
  IF[FIX[#4120/100] GT 5]GOTO 9; ..... Out–of–range tool number
  #[500+FIX[#4120/100]]=#3002+#[500+FIX[#4120/100]];
 ..... Calculates cumulative time.

N9 M05; ..... Stops the spindle.
  M99;
```

15.7 PROCESSING MACRO STATEMENTS

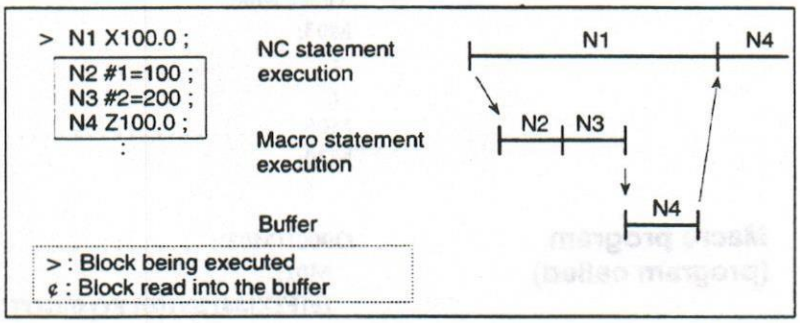
For smooth machining, the NC prereads the NC statement to be performed next. This operation is referred to as buffering. In tool nose radius compensation mode (G41, G42), the NC prereads NC statements two or three blocks ahead to find intersections. Macro statements for arithmetic expressions and conditional branches are processed as soon as they are read into the buffer. Blocks containing M00, M01, M02, or M30, blocks containing M codes for which buffering is suppressed by setting parameters 3411 to 3420, and blocks containing G31 are not preread.

Explanations

- **When the next block is not buffered (M codes that are not buffered, G31, etc.)**

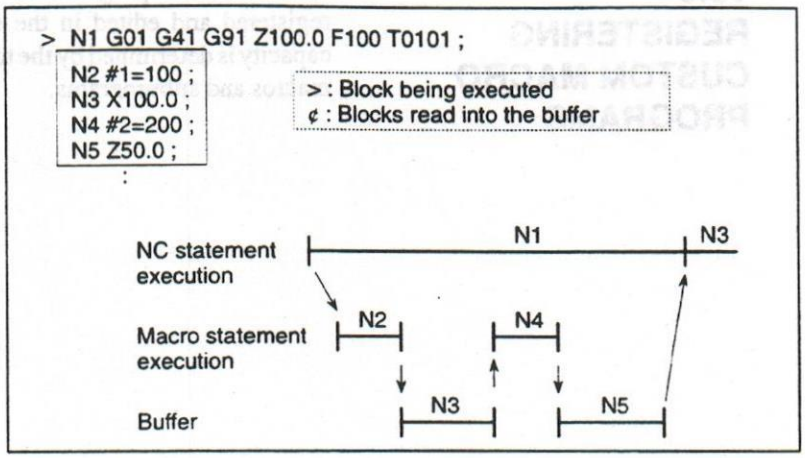


- **Buffering the next block in other than tool nose radius compensation mode (G41, G42) (normally prereading one block)**



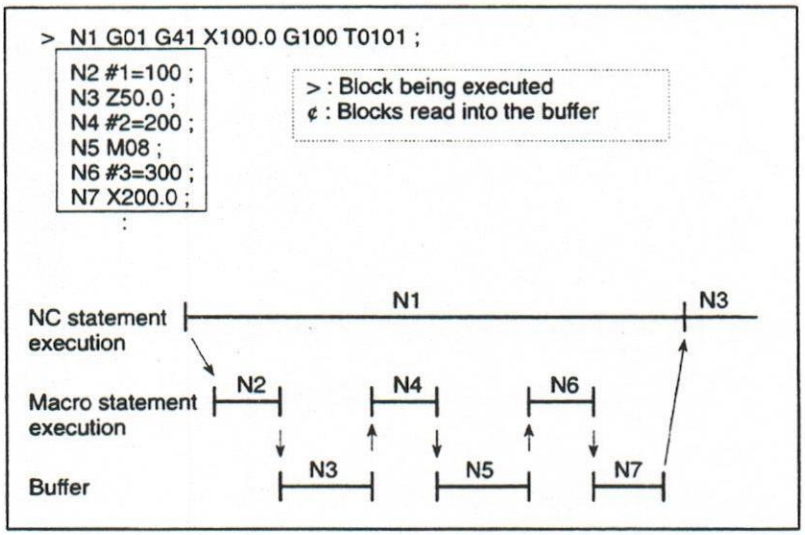
When N1 is being executed, the next NC statement (N4) is read into the buffer. The macro statements (N2, N3) between N1 and N4 are processed during execution of N1.

- **Buffering the next block in tool nose radius compensation mode (G41, G42)**



When N1 is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. The macro statements (N2, N4) between N1 and N5 are processed during execution of N1.

- **When the next block involves no movement in tool nose radius compensation C (G41, G42) mode**



When the NC1 block is being executed, the NC statements in the next two blocks (up to N5) are read into the buffer. Since N5 is a block that involves no movement, an intersection cannot be calculated. In this case, the NC statements in the next three blocks (up to N7) are read. The macro statements (N2, N4, and N6) between N1 and N7 are processed during execution of N1.

15.8 REGISTERING CUSTOM MACRO PROGRAMS

Custom macro programs are similar to subprograms. They can be registered and edited in the same way as subprograms. The storage capacity is determined by the total length of tape used to store both custom macros and subprograms.

15.9 LIMITATIONS

- **MDI operation**

  The macro call command can be specified in MDI mode too. During automatic operation, however, it is impossible to switch to the MDI mode for a macro program call.

- **Sequence number search**

  A custom macro program cannot be searched for a sequence number.

- **Single block**

  Even while a macro program is being executed, blocks can be stopped in the single block mode (except blocks containing macro call commands, arithmetic operation commands, and control commands).
  A block containing a macro call command (G65, G66, or G67) does not stop even when the single block mode is on. Blocks containing arithmetic operation commands and control commands can be stopped in single block mode by setting SBM (bit 5 of parameter 6000) to 1.
  Single block stop operation is used for testing custom macro programs. Note that when a single block stop occurs at a macro statement in tool nose radius compensation mode, the statement is assumed to be a block that does not involve movement, and proper compensation cannot be performed in some cases. (Strictly speaking, the block is regarded as specifying a movement with a travel distance 0.)

- **Optional block skip**

  A / appearing in the middle of an <expression> (enclosed in brackets [ ] on the right-hand side of an arithmetic expression) is regarded as a division operator; it is not regarded as the specifier for an optional block skip code.

- **Operation in EDIT mode**

  Registered custom macro programs and subprograms should be protected from being destroyed by accident. By setting NE8 (bit 0 of parameter 3202) and NE9 (bit 4 of parameter 3202) to 1, deletion and editing are disabled for custom macro programs and subprograms with program numbers 8000 to 8999 and 9000 to 9999. When the entire memory is cleared (by pressing the 7 and 9 keys at the same time to turn on the power), the contents of memory such as custom macro programs are deleted.

- **Reset**

  When memory is cleared with a reset operation, local variables and common variables #100 to #149 are cleared to null values. They can be prevented from being cleared by setting, CLV and CCV (bits 7 and 6 of parameter 6001). System variables #1000 to #1133 are not cleared.
  A reset operation clears any called states of custom macro programs and subprograms, and any DO states, and returns control to the main program.

- **Display of the PROGRAM RESTART page**

  As with M98, the M and T codes used for subprogram calls are not displayed.

- **Feed hold**

  When a feed hold is enabled during execution of a macro statement, the machine stops after execution of the macro statement. The machine also stops when a reset or alarm occurs.

- **Constant values that can be used in <expression>**

  +0.0000001 to +99999999
  –99999999 to –0.0000001
  The number of significant digits is 8 (decimal). If this range is exceeded, alarm No. 003 occurs.

15.10 EXTERNAL OUTPUT COMMANDS

In addition to the standard custom macro commands, the following macro commands are available. They are referred to as external output commands.

- **BPRNT**
- **DPRNT**
- **POPEN**
- **PCLOS**

These commands are provided to output variable values and characters through the reader/punch interface.

Explanations

Specify these commands in the following order:

Open command: POPEN
Before specifying a sequence of data output commands, specify this command to establish a connection to an external input/output device.

Data output command: BPRNT or DPRNT
Specify necessary data output.

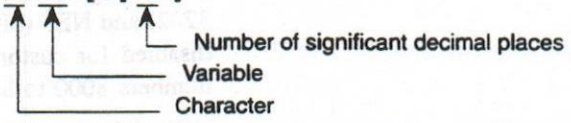
Close command: PCLOS
When all data output commands have completed, specify PCLOS to release a connection to an external input/output device.

- **Open command POPEN**

POPEN
POPEN establishes a connection to an external input/output device. It must be specified before a sequence of data output commands. The NC outputs a DC2 control code.

- **Data output command BPRNT**

BPRNT [a #b [c] ...]



The BPRNT command outputs characters and variable values in binary.

(i) Specified characters are converted to corresponding codes according to the setting data (ISO) that is output at that time.
Specifiable characters are as follows:
- **Letters (A to Z)**
- **Numbers**
- **Special characters (*, /, +, –, etc.)**

An asterisk (*) is output by a space code.

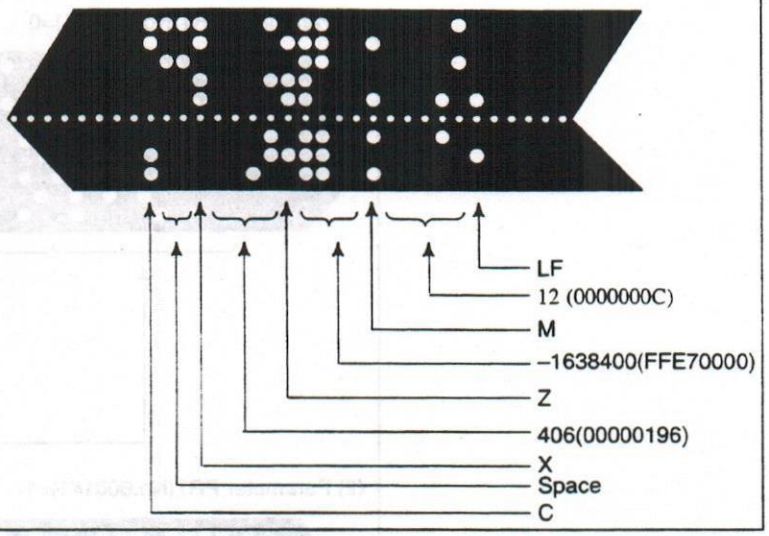
(ii) All variables are stored with a decimal point. Specify a variable followed by the number of significant decimal places enclosed in brackets. A variable value is treated as 2–word (32–bit) data, including the decimal digits. It is output as binary data starting from the highest byte.

(iii) When specified data has been output, an EOB code is output according to the ISO code settings on the parameter screen.

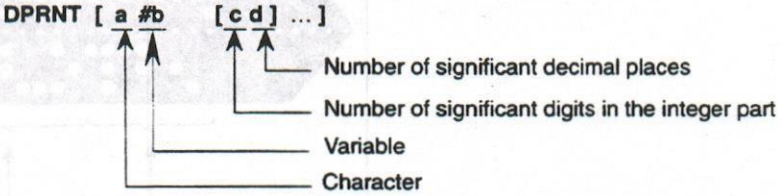
(iv) Null variables are regarded as 0.

Example)

BPRINT [C X#100 [3] Z#101 [3] M#10 [0]]**
Variable value
#100=0.40596
#101=–1638.4
#10=12.34



- **Data output command DPRNT**



The DPRNT command outputs characters and each digit in the value of a variable according to the code set in the settings (ISO).

(i) For an explanation of the DPRNT command, see Items (i), (iii), and (iv) for the BPRNT command.

(ii) When outputting a variable, specify # followed by the variable number, then specify the number of digits in the integer part and the number of decimal places enclosed in brackets.
One code is output for each of the specified number of digits, starting with the highest digit. For each digit, a code is output according to the settings (ISO). The decimal point is also output using a code set in the settings (ISO).
Each variable must be a numeric value consisting of up to eight digits. When high–order digits are zeros, these zeros are not output if PRT (bit1 of parameter 6001) is 1. If PRT is 0, a space code is output each time a zero is encountered.
When the number of decimal places is not zero, digits in the decimal part are always output. If the number of decimal places is zero, no decimal point is output.
When PRT (bit 1 of parameter 6001) is 0, a space code is output to indicate a positive number instead of +; if PRT is 1, no code is output.

Example)

DPRNT [X#2 [53] Z#5 [53] T#30 [20]]

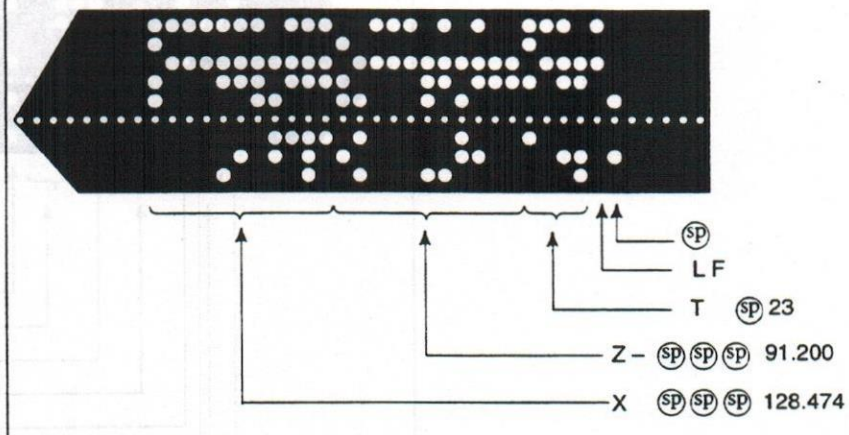
Variable value

#2=128.47398

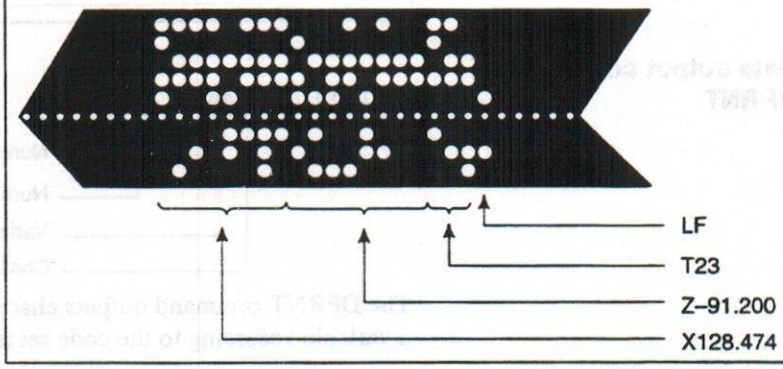
#5=-91.2

#30=123.456

(1) Parameter PRT(No.6001#1)=0



(2) Parameter PRT(No.6001#1)=1



- **Close command PCLOS**

PCLOS ;

The PCLOS command releases a connection to an external input/output device. Specify this command when all data output commands have terminated. DC4 control code is output from the CNC.

- **Required setting**

Specify the channel use for setting data (I/O channel). According to the specification of this parameter, set data items (such as the baud rate) for the reader/punch interface.

I/O channel 0 : Parameters 101, 102, and 103
I/O channel 1 : Parameters 111, 112, and 113

Specify parameter 102, 112 or 122 so that the reader/punch interface is used as the output device for punching. (Never specify output to the Fanuc Cassette or floppy disks.)

When specifying a DPRNT command to output data, specify whether leading zeros are output as spaces (by setting PRT (bit 1 of parameter 6001) to 1 or 0).

To indicate the end of a line of data in ISO code, specify whether to use only an LF (CRO, of bit 4 of parameter 6001 is 0) or an LF and CR (CRO is 1).

Notes

Notes

1 It is not necessary to always specify the open command (POPEN), data output command (BPRNT, DPRNT), and close command (PCLOS) together. Once an open command is specified at the beginning of a program, it does not need to be specified again except after a close command was specified.
2 Be sure to specify open commands and close commands in pairs. Specify the close command at the end of the program. However, do not specify a close command if no open command has been specified.
3 When a reset operation is performed while commands are being output by a data output command, output is stopped and subsequent data is erased. Therefore, when a reset operation is performed by a code such as M30 at the end of a program that performs data output, specify a close command at the end of the program so that processing such as M30 is not performed until all data is output.
4 Abbreviated macro words enclosed in brackets [ ] remains unchanged. However, note that when the characters in brackets are divided and input several times, the second and subsequent abbreviations are converted and input.
5 O can be specified in brackets [ ]. Note that when the characters in brackets [ ] are divided and input several times, O is omitted in the second and subsequent inputs.

16 PROGRAMMABLE PARAMETER ENTRY (G10)

General

The values of parameters can be entered in a program. This function is used for setting pitch error compensation data when attachments are changed or the maximum cutting feedrate or cutting time constants are changed to meet changing machining conditions.

Format

Format
G10L50; Parameter entry mode setting **N_R_;** For parameters other than the axis type **N_P_R_;** For axis type parameters : **G11;** Parameter entry mode cancel
Meaning of command
N_: Parameter No. (4digits) or compensation position No.(0 to 1023) for pitch errors compensation +10,000 (5digit) **R_:** Parameter setting value (Leading zeros can be omitted.) **P_:** Axis No. 1 to 2 (Used for entering axis type parameters)

Explanations

- **Parameter setting value (R_)**

Do not use a decimal point in a value set in a parameter (R_).
a decimal point cannot be used in a custom macro variable for R_ either.

- **Axis No. (P_)**

Specify an axis number (P_) from 1 to 2 (up to two axes) for an axis type parameter. The control axes are numbered in the order in which they are displayed on the CNC display.
For example, specify P2 for the control axis which is displayed second.

Notes

Do not fail to perform reference point return manually after changing the pitch error compensation data or backlash compensation data. Without this, the machine position can deviate from the correct position.
Other NC statements cannot be specified while in parameter input mode.
The canned–cycle mode must be cancelled before entering of parameters. When not cancelled, the channed–cycle motion will be activated.

Examples

1 Set bit 2 (SPB) of bit type parameter No.3404.

```
G10L50 ;                Parameter entry mode
N3404 R 00000100 ;      SBP setting
G11 ;                   cancel parameter entry mode
```

2 Change the values for the X–axis and Z–axis in axis type parameter No.1321 (the coordinates of stored stroke limit 1 in the positive direction for each axis).

```
G10L50 ;                Parameter entry mode
N1321P1R4500 ;          Modify X axis
N1321P2R12000 ;         Modify Z axis
G11 ;                   cancel parameter entry mode
```

III. OPERATION

1 GENERAL

1.1 MANUAL OPERATION

Explanations

- **Manual reference position return (See Section III-3.1)**

The CNC machine tool has the position only for the machine.
This position is called the reference position, where the tool is replaced or the coordinate are set. Ordinarily, after the power is turned on, the tool is moved to the reference position.
Manual reference position return is to move the tool to the reference position using switches and pushbuttons located on the operator's panel.

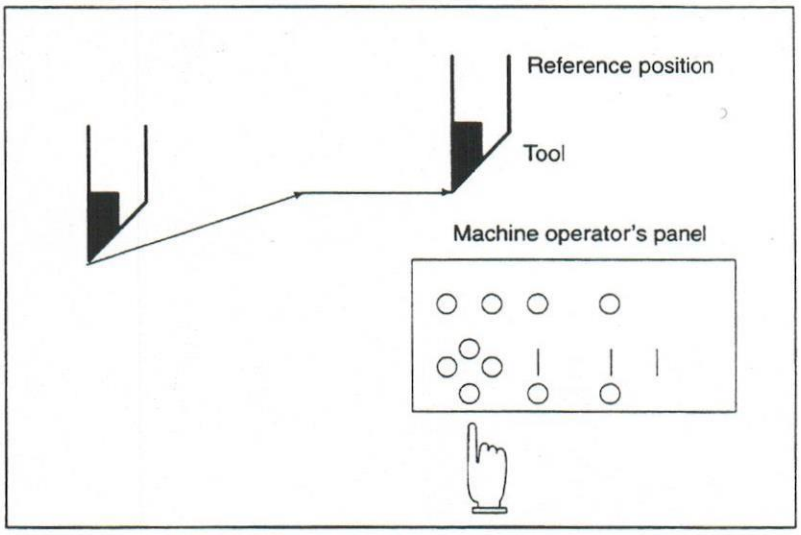


Fig.1.1 (a) Manual reference position return

The tool can be moved to the reference position also with program commands.
This operation is called automatic reference position return (See Section II-6).

• The tool movement by manual operation

Using machine operator's panel switches, push buttons, or the manual handle, the tool can be moved along each axis.

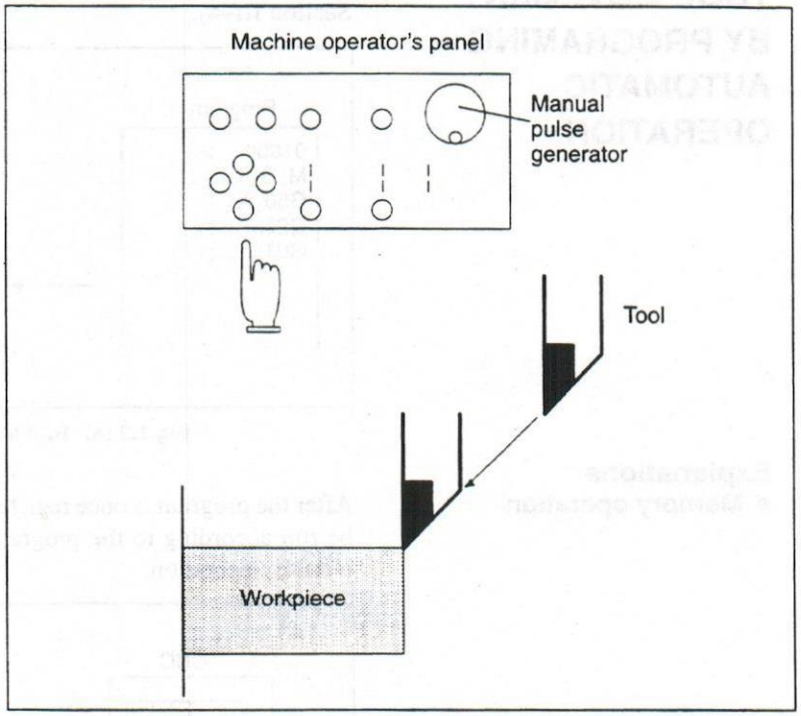


Fig.1.1 (b) The tool movement by manual operation

The tool can be moved in the following ways:

(i)Jog feed (See Section III–3.2)

The tool moves continuously while a push button remains pressed.

(ii)Incremental feed (See Section III–3.3)

The tool moves by the predetermined distance each time a button is pressed.

(iii)Manual handle feed (See Section III–3.4)

By rotating the manual handle,the tool moves by the distance corresponding to the degree of handle rotation.

1.2 TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION

Automatic operation is to operate the machine according to the created program. It includes memory and MDI, and DNC operations. (See Section III–4).

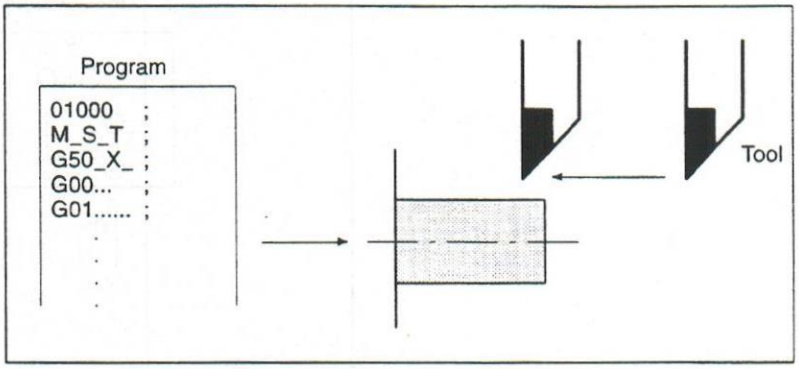


Fig.1.2 (a) Tool Movement by Programming

Explanations

- **Memory operation**

After the program is once registered in memory of CNC, the machine can be run according to the program instructions. This operation is called memory operation.

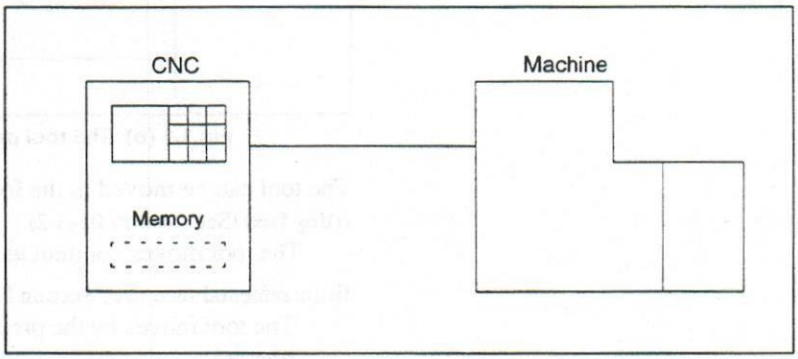


Fig.1.2 (b) Memory Operation

- **DNC operation**

The machine can operate by reading programs directly from external input/output devices without having to register those programs in CNC memory. This is called DCN operation.

- **MDI operation**

After the program is entered, as an command group, from the MDI keyboard, the machine can be run according to the program. This operation is called MDI operation.

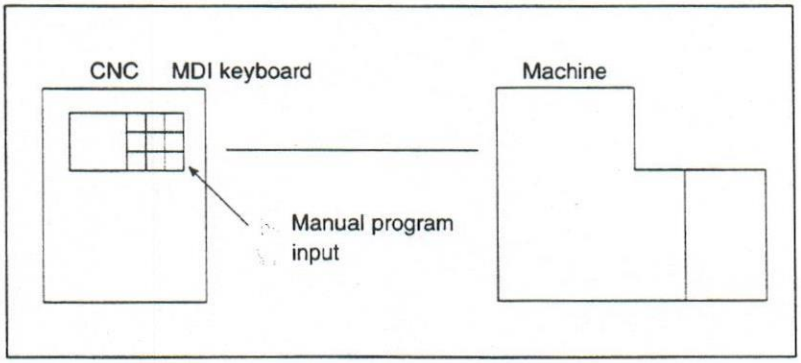


Fig.1.2 (c) MDI operation

1.3 AUTOMATIC OPERATION

Explanations

- **Program selection**

Select the program used for the workpiece. Ordinarily, one program is prepared for one workpiece. If two or more programs are in memory, select the program to be used, by searching the program number (Section III–9.4).

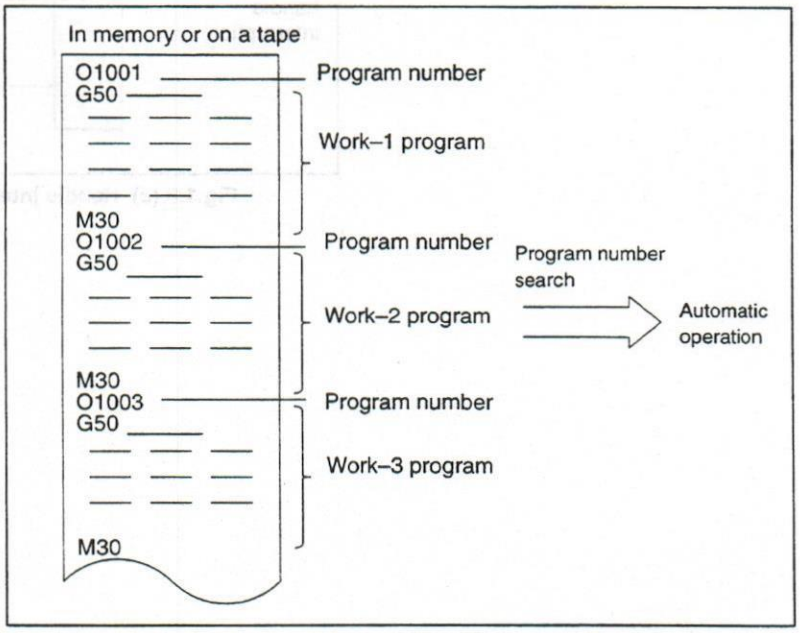


Fig.1.3 (a) Program Selection for Automatic Operation

- **Start and stop (See Section III–4)**

Pressing the cycle start push button causes automatic operation to start. By pressing the feed hold or reset push button, automatic operation pauses or stops. By specifying the program stop or program termination command in the program, the running will stop during automatic operation. When one process machining is completed, automatic operation stops.

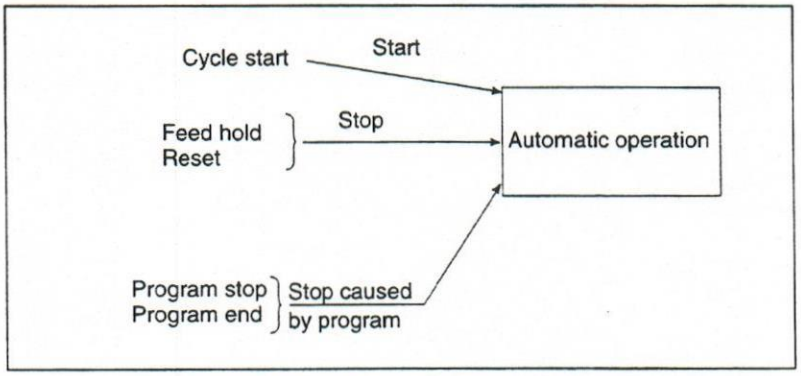


Fig.1.3 (b) Start and Stop for Automatic Operation

- **Handle interruption (See Section III–4.6)**

While automatic operation is being executed, tool movement can overlap automatic operation by rotating the manual handle.

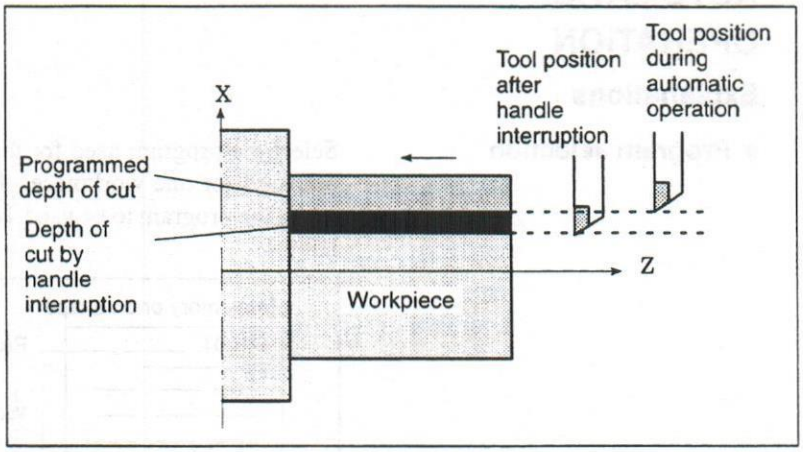


Fig.1.3 (c) Handle Interruption for Automatic Operation

1.4 TESTING A PROGRAM

Before machining is started, the automatic running check can be executed. It checks whether the created program can operate the machine as desired. This check can be accomplished by running the machine actually or viewing the position display change (without running the machine) (See Section III–5).

1.4.1 Check by Running the Machine

Explanations

- **Dry run (See Section III–5.4)**

Remove the workpiece, check only movement of the tool. Select the tool movement rate using the dial on the operator's panel.

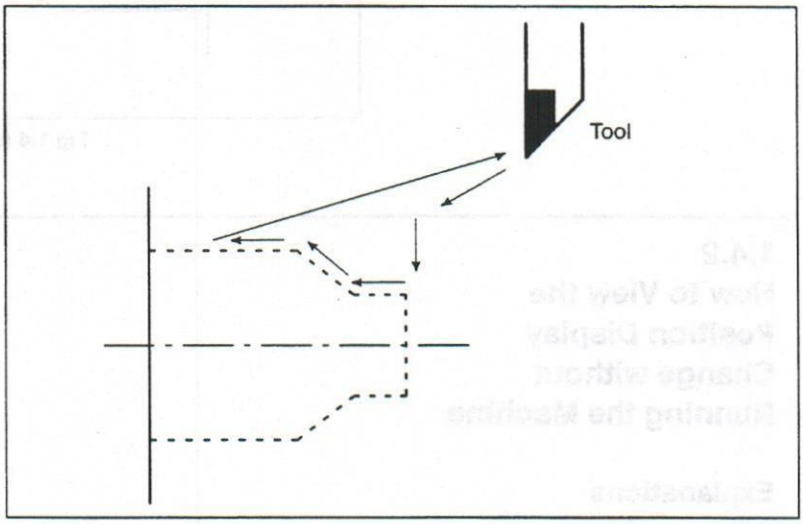


Fig.1.4 (a) Dry run

- **Feedrate override (See Section III–5.2)**

Check the program by changing the feed rate specified in the program.

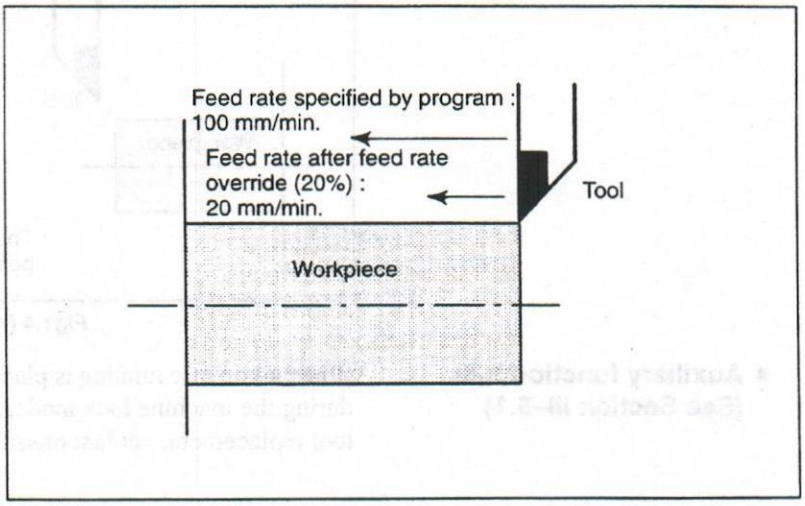


Fig1.4 (b) Feedrate Override

- **Single block (See Section III–5.5)**

When the cycle start push button is pressed, the tool executes one operation then stops. By pressing the cycle start again, the tool executes the next operation then stops. The program is checked in this manner.

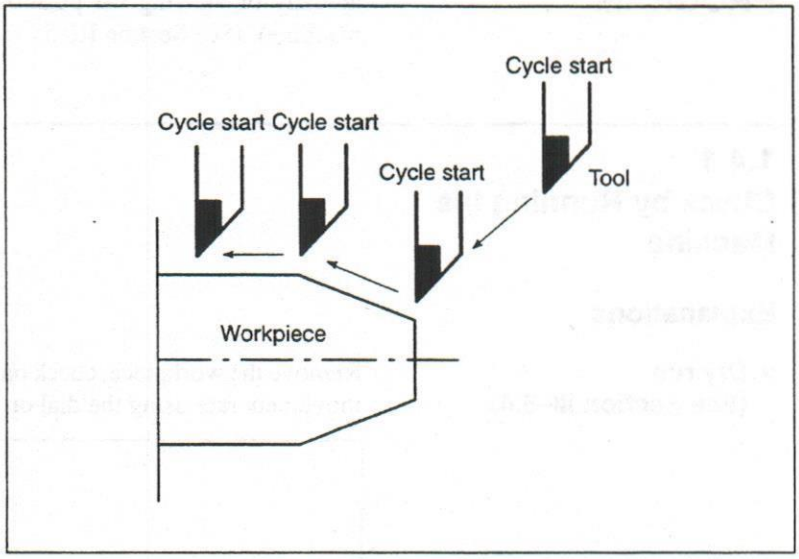


Fig.1.4 (c) Single Block

1.4.2 How to View the Position Display Change without Running the Machine

Explanations

- **Machine lock (See Sections III–5.1)**

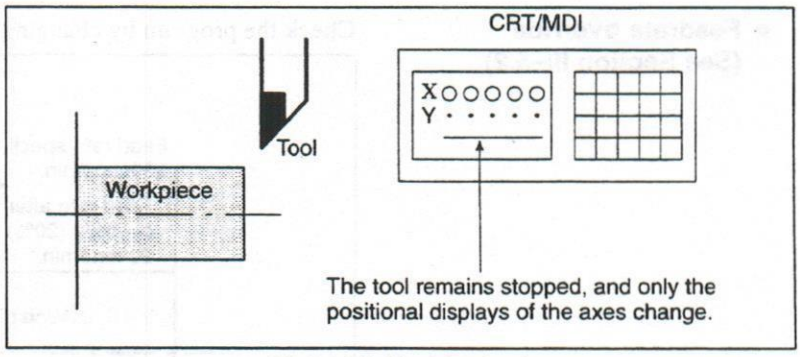


Fig1.4 (d) Machine Lock

- **Auxiliary function lock (See Section III–5.1)**

When automatic running is placed into the auxiliary function lock mode during the machine lock mode, all auxiliary functions (spindle rotation, tool replacement, coolant on/off, etc.) are disabled.

1.5 EDITING A PART PROGRAM

After a created program is once registered in memory, it can be corrected or modified from the CRT/MDI panel (See Section III–9).
This operation can be executed using the part program storage/edit function.

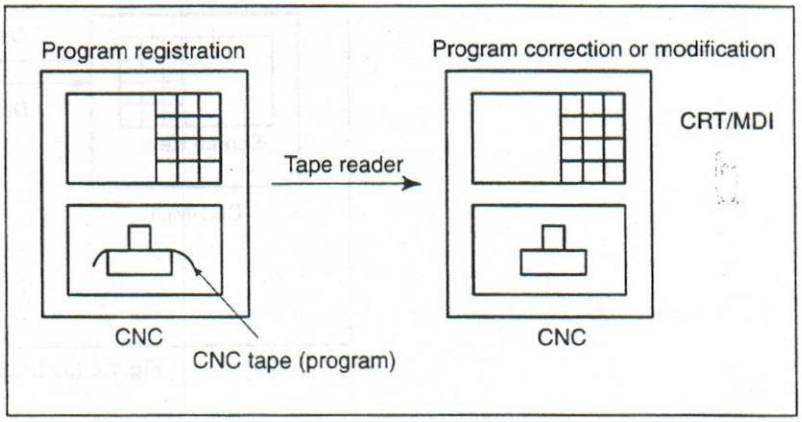


Fig.1.5 (a) Part Program Editing

1.6 DISPLAYING AND SETTING DATA

The operator can display or change a value stored in CNC internal memory by key operation on the CRT/MDI screen (See III–11).

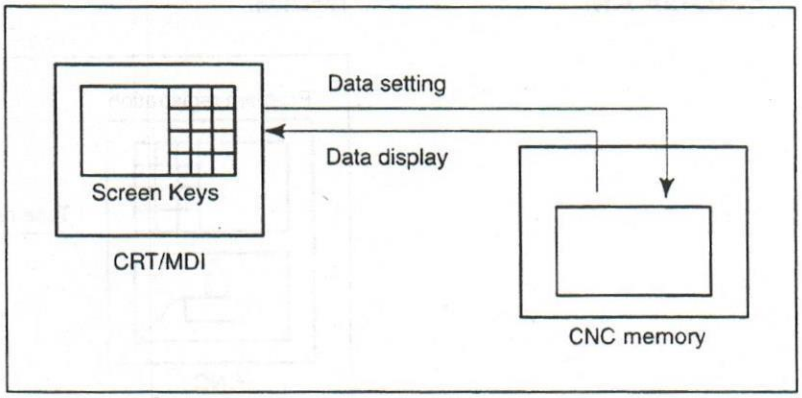


Fig.1.6 (a) Displaying and Setting Data

Explanations

- **Offset value**

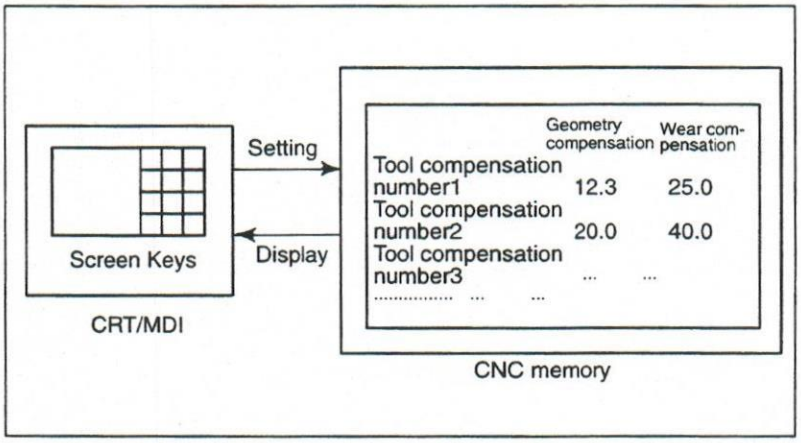


Fig.1.6 (b) Displaying and Setting Offset Values

The tool has the tool dimension (length, diameter). When a workpiece is machined, the tool movement value depends on the tool dimensions. By setting tool dimension data in CNC memory beforehand, automatically generates tool routes that permit any tool to cut the workpiece specified by the program. Tool dimension data is called the offset value (See Section III–11.4.1).

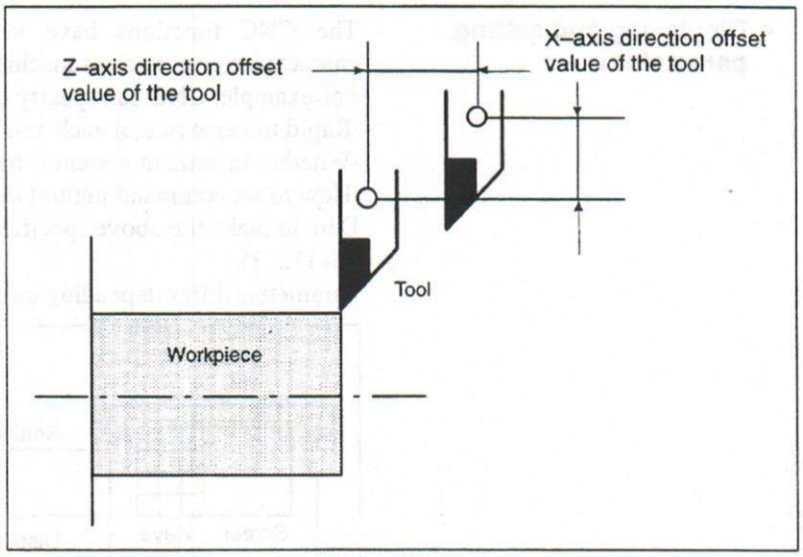


Fig.1.6 (c) Offset Value

- **Displaying and setting operator's setting data**

Apart from parameters, there is data that is set by the operator in operation. This data causes machine characteristics to change.
For example, the following data can be set:
·Inch/Metric switching
·I/O devices selecting
·Mirror image cutting on/off
The above data is called setting data (See Section III–11.4.5).

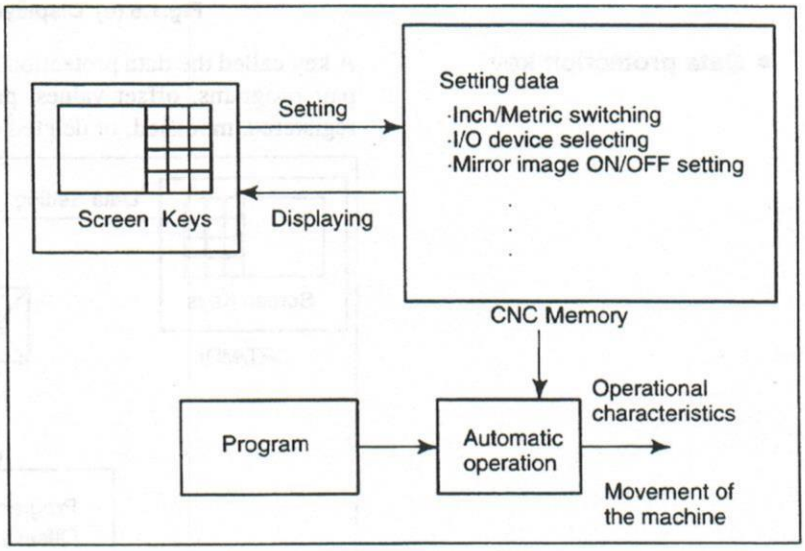


Fig.1.6 (d) Displaying and Setting Operator's setting data

- **Displaying and setting parameters**

The CNC functions have versatility in order to take action in characteristics of various machines.
For example, CNC can specify the following:
·Rapid traverse rate of each axis
·Whether increment system is based on metric system or inch system.
·How to set command multiply/detect multiply (CMR/DMR)
Data to make the above specification is called parameters (See Section III–11.5.1).
Parameters differ depending on machine tool.

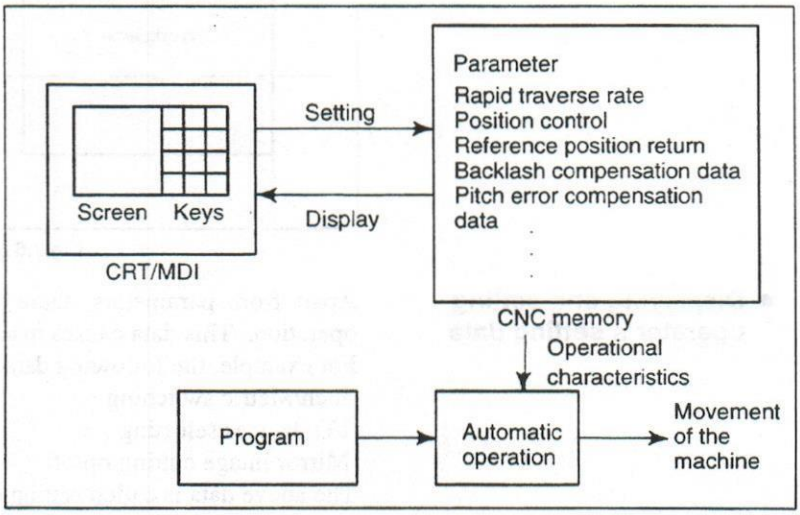


Fig.1.6 (e) Displaying and setting parameters

- **Data protection key**

A key called the data protection key can be defined. It is used to prevent part programs, offset values, parameters, and setting data from being registered, modified, or deleted erroneously (See Section III–11).

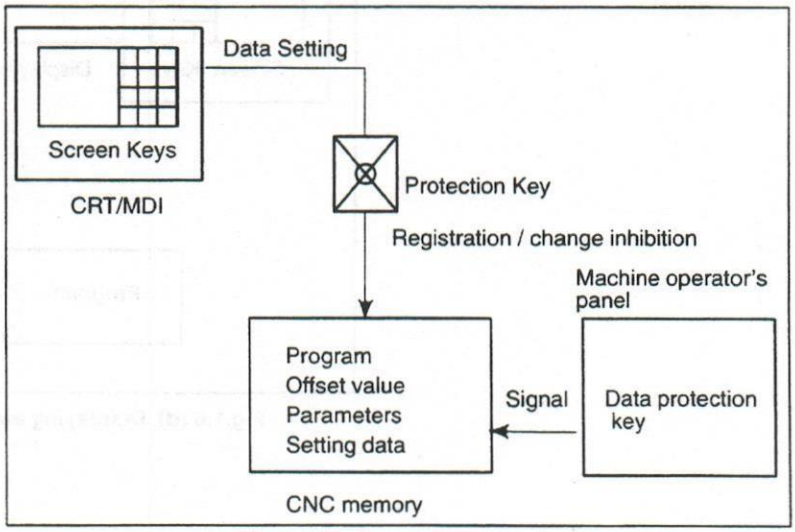
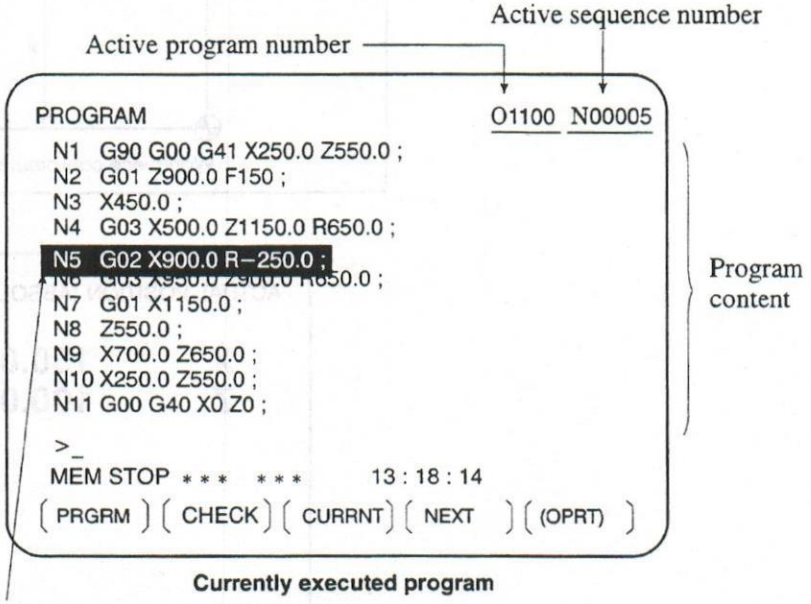


Fig.1.6 (f) Data Protection Key

1.7 DISPLAY

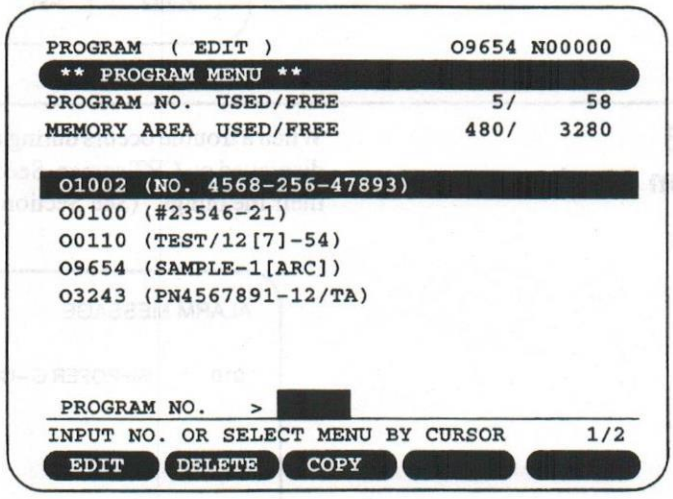
1.7.1 Program Display

The contents of the currently active program are displayed. In addition, the programs scheduled next and the program list are displayed. (See Section III–11.2, III–11.3)


Active sequence number
Active program number
PROGRAM O1100 N00005
N1 G90 G00 G41 X250.0 Z550.0 ;
N2 G01 Z900.0 F150 ;
N3 X450.0 ;
N4 G03 X500.0 Z1150.0 R650.0 ;
N5 G02 X900.0 R–250.0 ;
N7 G01 X1150.0 ;
N8 Z550.0 ;
N9 X700.0 Z650.0 ;
N10 X250.0 Z550.0 ;
N11 G00 G40 X0 Z0 ;
>_
MEM STOP *** *** 13 : 18 : 14
[PRGRM] [CHECK] [CURRNT] [NEXT] [(OPRT)]
Program content

Currently executed program

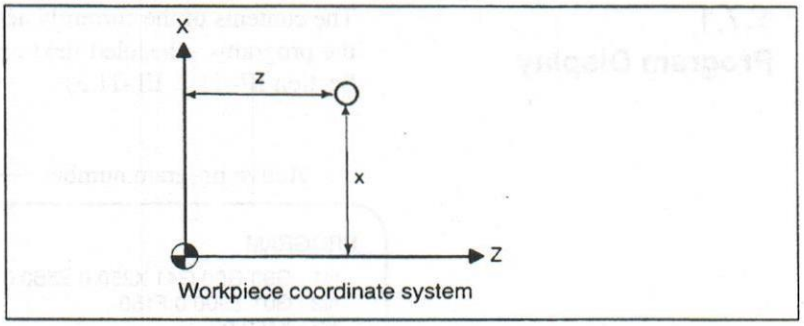
The cursor indicates the currently executed location



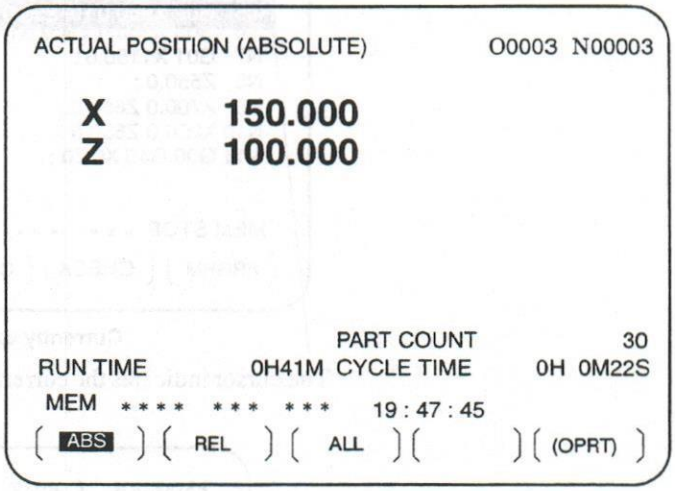
Program list

1.7.2 Current Position Display

The current position of the tool is displayed with the coordinate values. The distance from the current position to the target position can also be displayed. (See Section III–11.1 to 11.1.3)

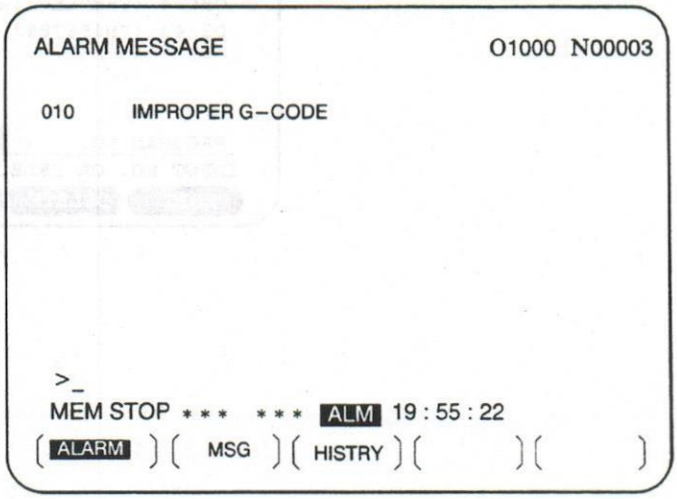


Workpiece coordinate system


ACTUAL POSITION (ABSOLUTE)
O0003 N00003
X 150.000
Z 100.000
PART COUNT 30
RUN TIME 0H41M CYCLE TIME 0H 0M22S
MEM **** *** *** 19 : 47 : 45
[ABS] [REL] [ALL] [ ] [(OPRT)]

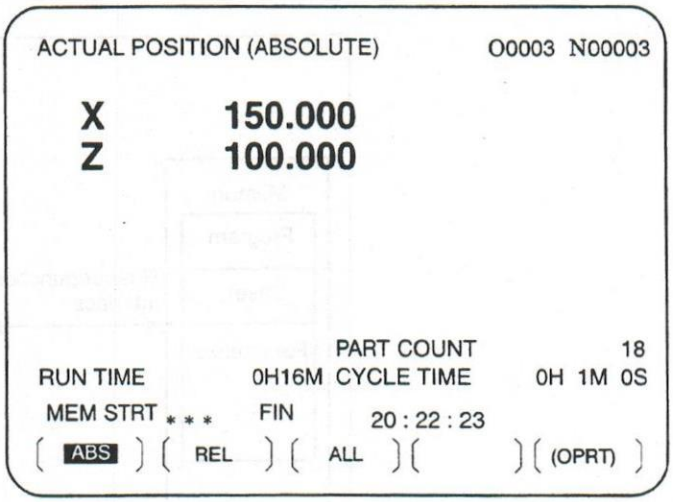
1.7.3 Alarm Display

When a trouble occurs during operation, error code and alarm message are displayed on CRT screen. See APPENDIX G for the list of error codes and their meanings. (See Section III–7.1)



1.7.4 Parts Count Display, Run Time Display

When this option is selected, two types of run time and number of parts are displayed on the screen. (See Section III–11.4.6)



1.8 DATA INPUT/OUTPUT

Programs, offset values, parameters, etc. input in CNC memory can be output to paper tape, cassette, or a floppy disk for saving. After once output to a medium, the data can be input into CNC memory. (See III–8)

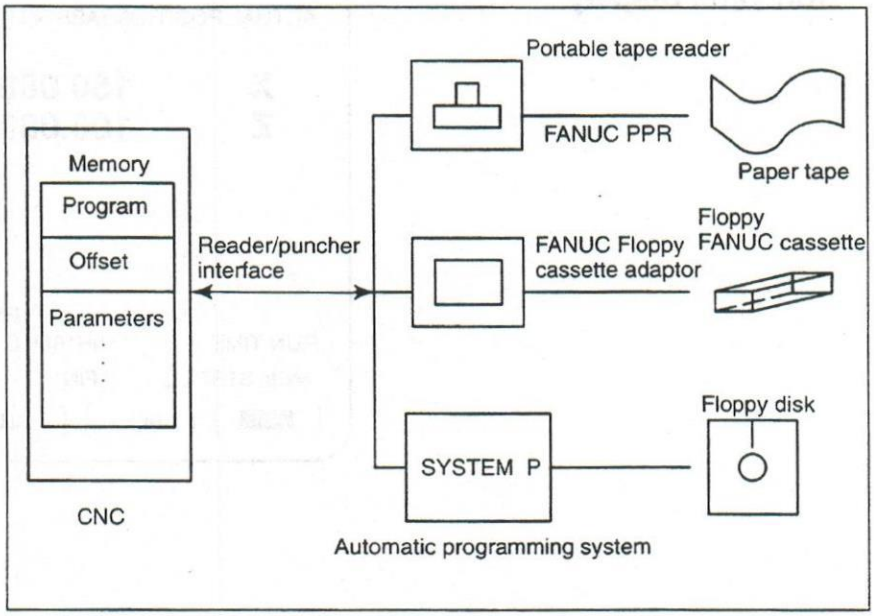


Fig.1.8 (a) Data Input/Output

2 OPERATIONAL DEVICES

The peripheral devices available include the CRT/MDI panel attached to the CNC, machine operator's panel and external input/output devices such as tape reader, PPR, floppy cassette, and FA card.

2.1 CRT/MDI PANELS AND LCD/MDI PANELS

Figs. 2.1 shows the CRT/MDI panel.

External view

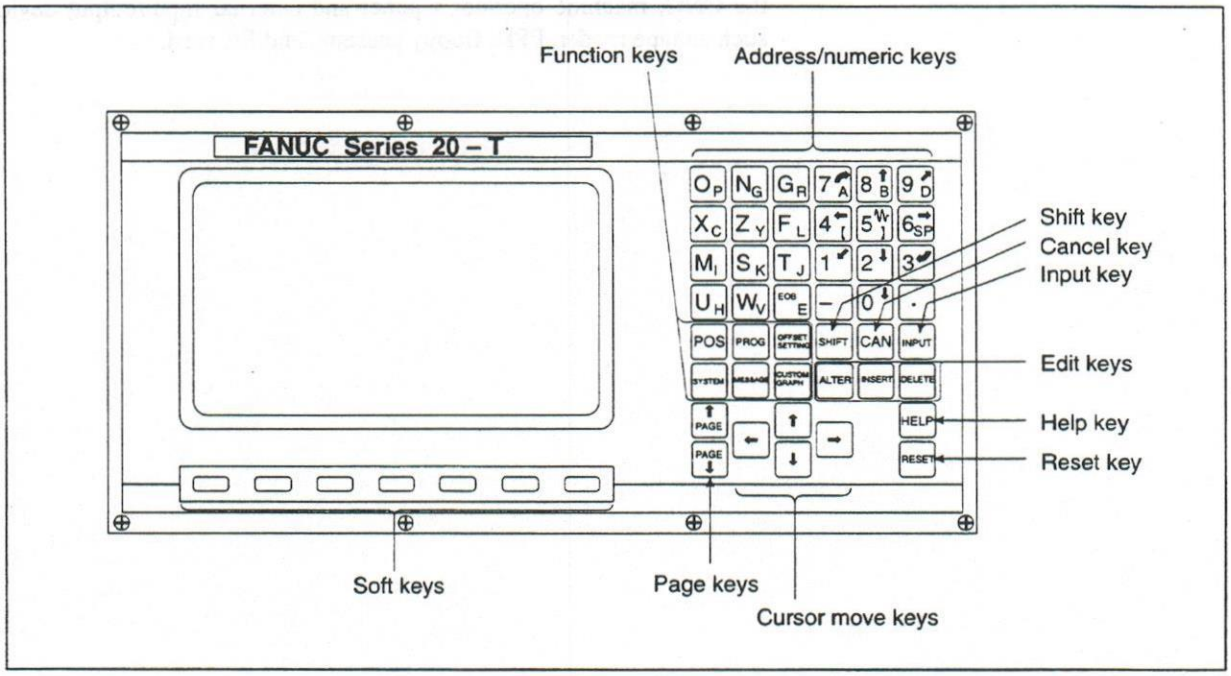


Fig2.1 9" small monochrome or color CRT/MDI (horizontal type)

Note

For the Series 20, an MDI created by the machine tool builder can be used. The key layout of such an MDI may not be as shown in Fig.2.1 (a). Refer to the relevant manual supplied by the machine tool builder.

Explanation of the keyboard

Table2.1 Explanation of the MDI keyboard

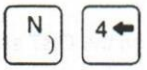
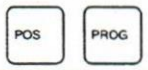
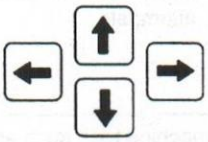
Number	Name	Explanation
1	RESET key	Press this key to reset the CNC, to cancel an alarm, etc.
2	HELP key	Press this key to display how to operate the machine tool, such as MDI key operation, or the details of an alarm which occurred in the CNC (Help function).
3	Soft keys	The soft keys have various functions, according to the Applications. The soft key functions are displayed at the bottom of the CRT screen.
4	Address and numeric keys  ...	Press these keys to input alphabetic, numeric, and other characters.
5	SHIFT key	Some keys have two characters on their keytop. Pressing the <SHIFT> key switches the characters. Special character Ê is displayed on the screen when a character indicated at the bottom right corner on the keytop can be entered.
6	INPUT key	When an address or a numerical key is pressed, the data is input to the buffer, and it is displayed on the CRT screen. To copy the data in the key input buffer to the offset register, etc., press the <INPUT> key. This key is equivalent to the [INPUT] key of the soft keys, and either can be pressed to produce the same result.
7	Cancel key	Press this key to delete the last character or symbol input to the key input buffer. When the key input buffer displays >N001X100Z_ and the cancel CAN key is pressed, Z is canceled and >N001X100_ is displayed.
8	Program edit keys	Press these keys when editing the program. ALTER : Alteration INSERT : Insertion DELETE : Deletion
9	Function keys  ...	Press theses keys to switch display screens for each function. See sec. 2.2 for details of the function keys.

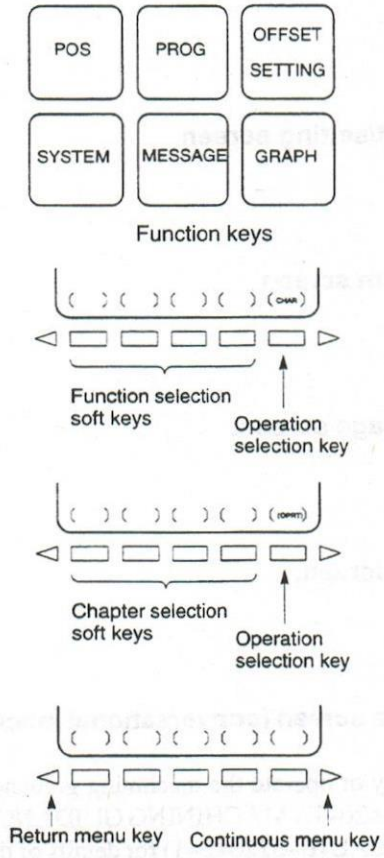
Table2.1 Explanation of the MDI keyboard

Number	Name	Explanation
10	Cursor move keys 	There are four different cursor move keys. : This key is used to move the cursor to the right or in the forward direction. The cursor is moved in short units in the forward direction. : This key is used to move the cursor to the left or in the reverse direction. The cursor is moved in short units in the reverse direction. : This key is used to move the cursor in a downward or forward direction. The cursor is moved in large units in the forward direction. : This key is used to move the cursor in an upward or reverse direction. The cursor is moved in large units in the reverse direction.
11	Page change keys	Two kinds of page change keys are described below. : This key is used to changeover the page on the CRT screen in the forward direction. : This key is used to changeover the page on the CRT screen in the reverse direction.

2.2 FUNCTION KEYS AND SOFT KEYS

Function keys are used to select the type of screen (function) to be displayed. After pressing a function key, you can then press a chapter selection soft key to select a subordinate screen (chapter) for that function. You can use function selection soft keys, instead of the function keys, to select screens (when bit 0 (FSK) of parameter No. 3101 is set to 0).

2.2.1 General Screen Operations



1 Press a function key on the CRT/MDI panel. The chapter selection soft keys that belong to the selected function appear.
After pressing a function selection soft key, you can then press the chapter selection key to display the chapter selection soft keys for that function (when bit 0 (FSK) of parameter No. 3101 is set to 0).

2 Press one of the chapter selection soft keys. The screen for the selected chapter appears. If the soft key for a target chapter is not displayed, press the continuous menu key (next–menu key).
In some cases, additional chapters can be selected within a chapter.

3 When the target chapter screen is displayed, press the operation selection key to display data to be manipulated.

4 To redisplay the chapter selection soft keys, press the return menu key.
When bit 0 (FSK) of parameter No. 3101 is set to 0, pressing the return menu key displays the function selection soft keys.

The general screen display procedure is explained above. However, the actual display procedure varies from one screen to another. For details, see the description of individual operations.

Notes
This manual describes how to select screens by using the function keys. The selection procedure is the same regardless of whether you use the function keys or the function selection soft keys.

2.2.2 Function Keys

Function keys are provided to select the type of screen to be displayed. The following function keys are provided on the CRT/MDI and LCD/MDI panels:

Press this key to display the **position screen**.

Press this key to display the **program screen**.

Press this key to display the **offset/setting screen**.

Press this key to display the **system screen**.

Press this key to display the **message screen.**

Press this key to display the **help screen.**

Press this key to display the **custom screen (conversational macro screen)**.
(Select the custom screen to display or operate the machining guidance function. Refer to the FANUC Series 20–TA MACHINING GUIDANCE FUNCTION OPERATOR'S MANUAL (B–62204E–1) for details of the machining guidance function. You can also use the custom screen to display an original screen created using the machine tool builder. For details, refer to the relevant manual supplied by the machine tool builder.)

2.2.3 Soft Keys

To display a more detailed screen, press a function key followed by a soft key. Soft keys are also used for actual operations.
The following illustrates how soft key displays are changed by pressing each function key.

The symbols in the following figures mean as shown below :

: Indicates screens

 : Indicates a screen that can be displayed by pressing a function key(*1)

[ ] : Indicates a soft key(*2)

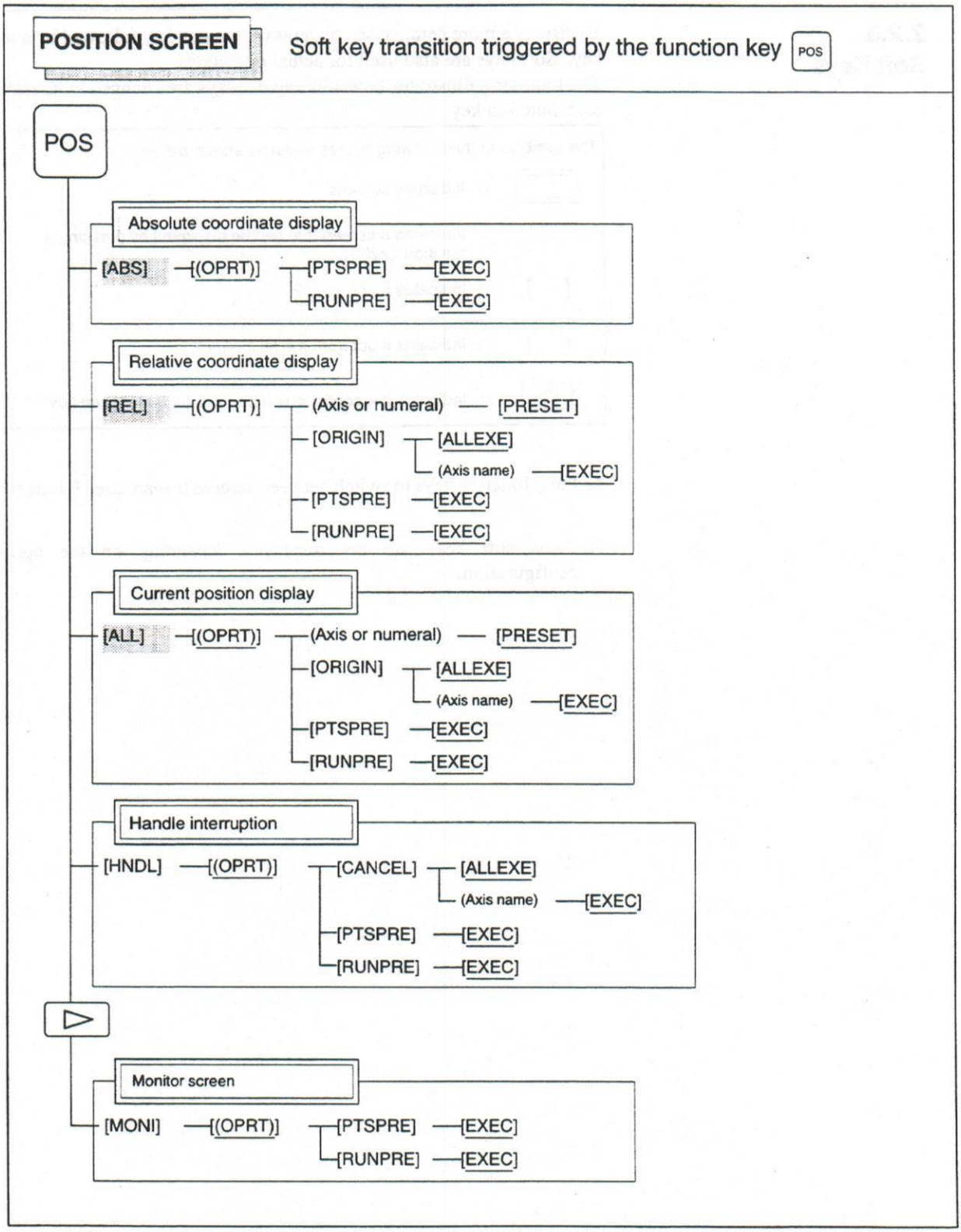
( ) : Indicates input from the MDI panel.

: Indicates the continuous menu key (rightmost soft key).

*1 Press function keys to switch between screens that are used frequently.

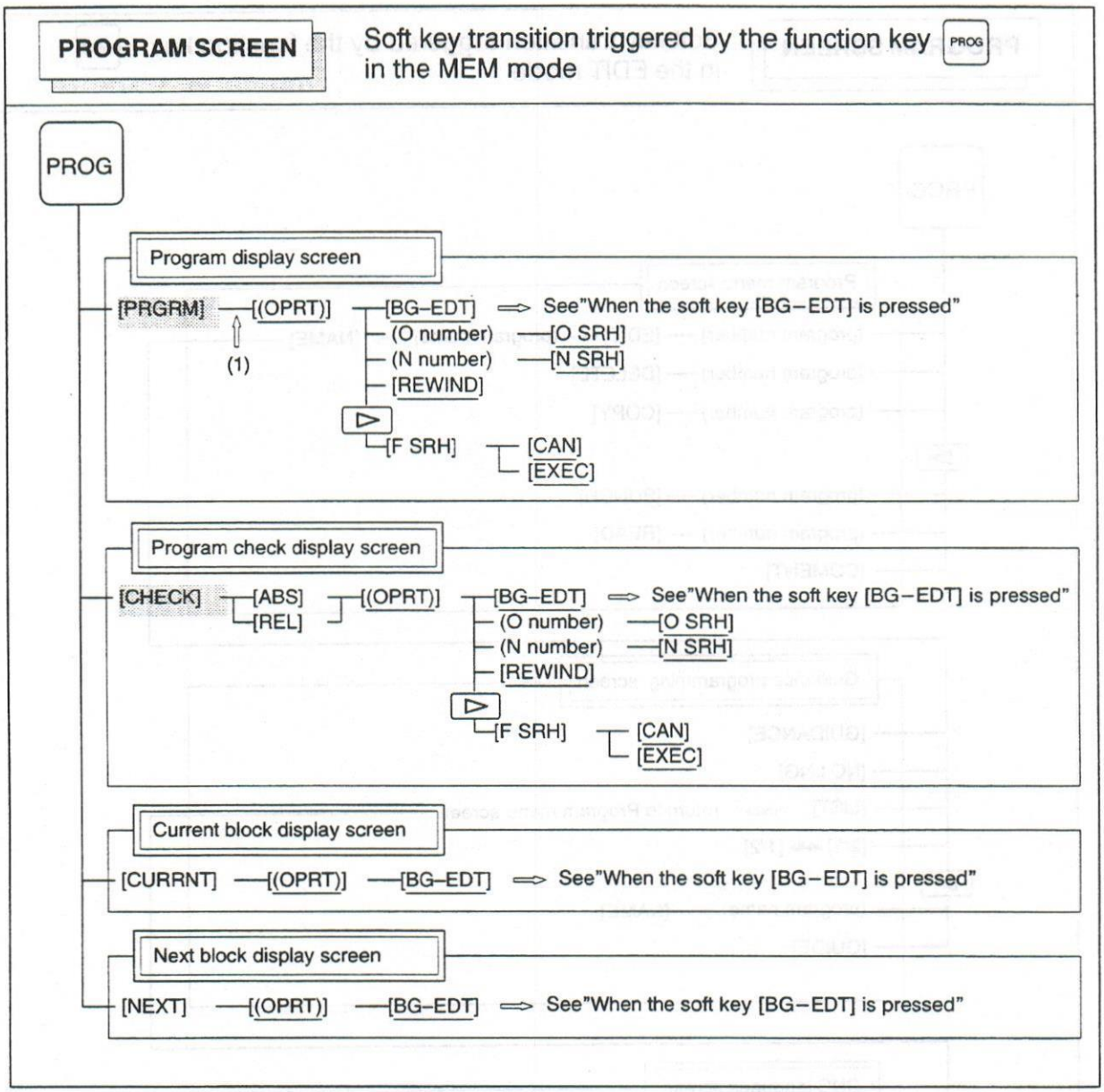
*2 Some soft keys are not displayed depending on the option configuration.

POSITION SCREEN Soft key transition triggered by the function key [POS]

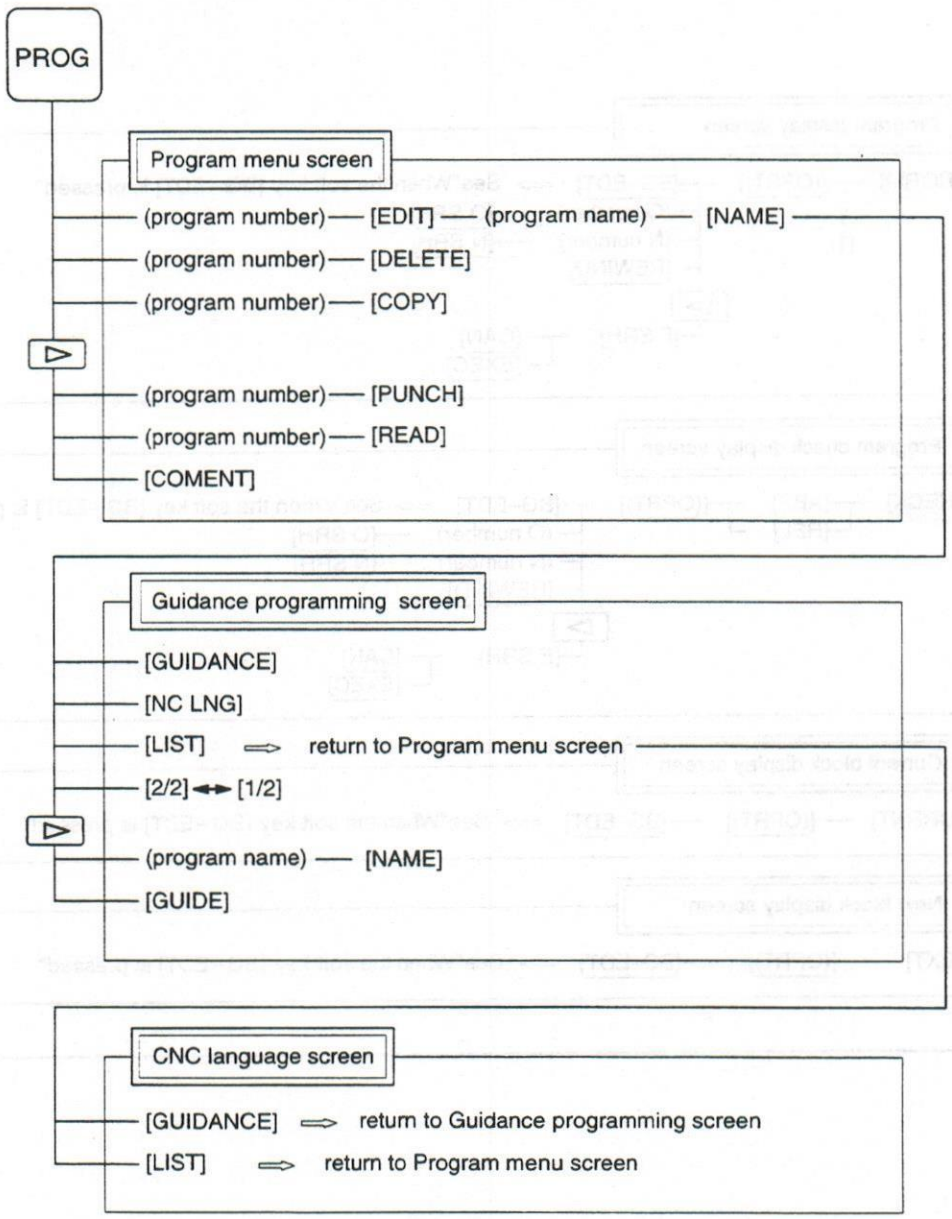


PROGRAM SCREEN

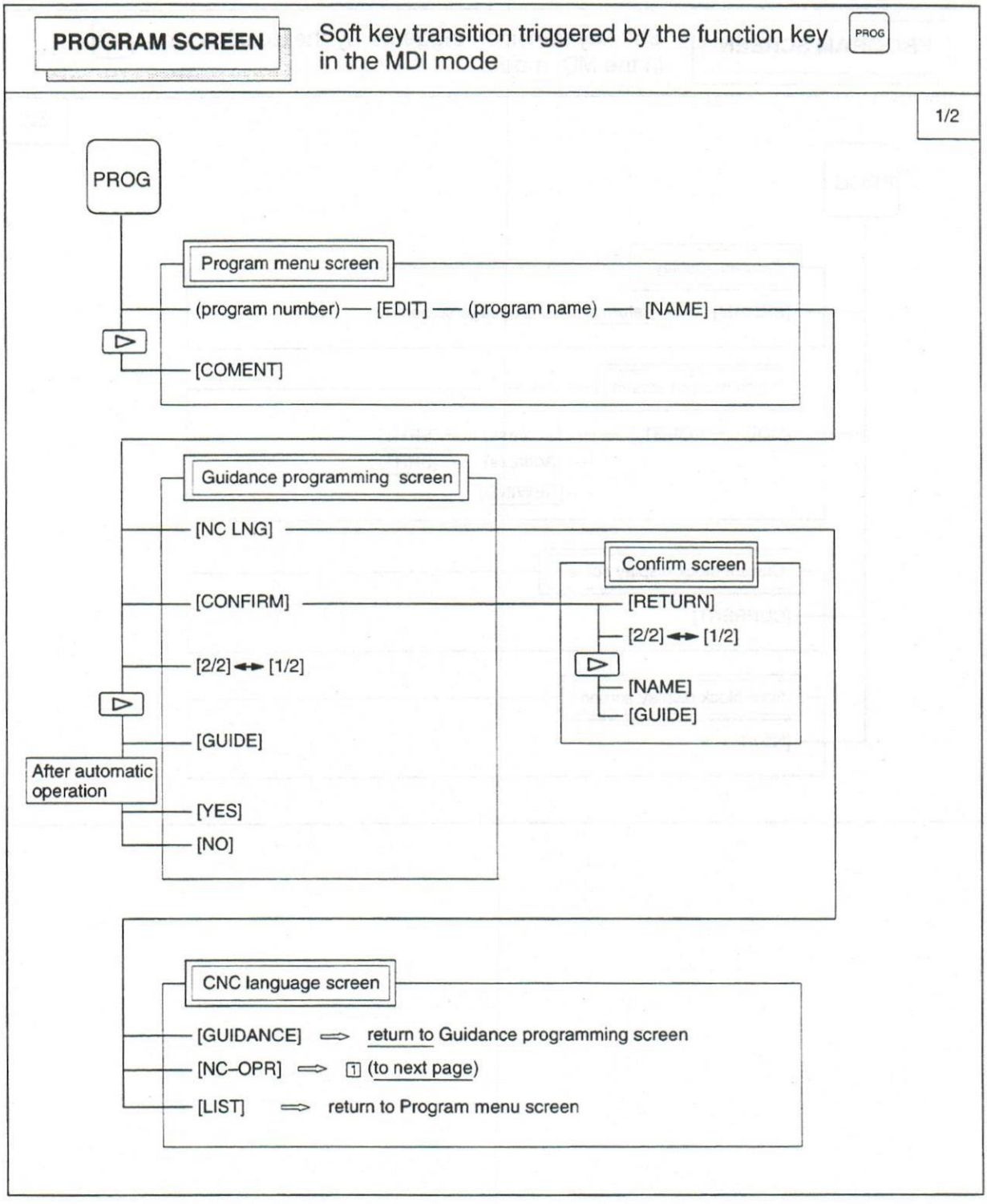
Soft key transition triggered by the function key PROG in the MEM mode



PROGRAM SCREEN Soft key transition triggered by the function key in the EDIT mode

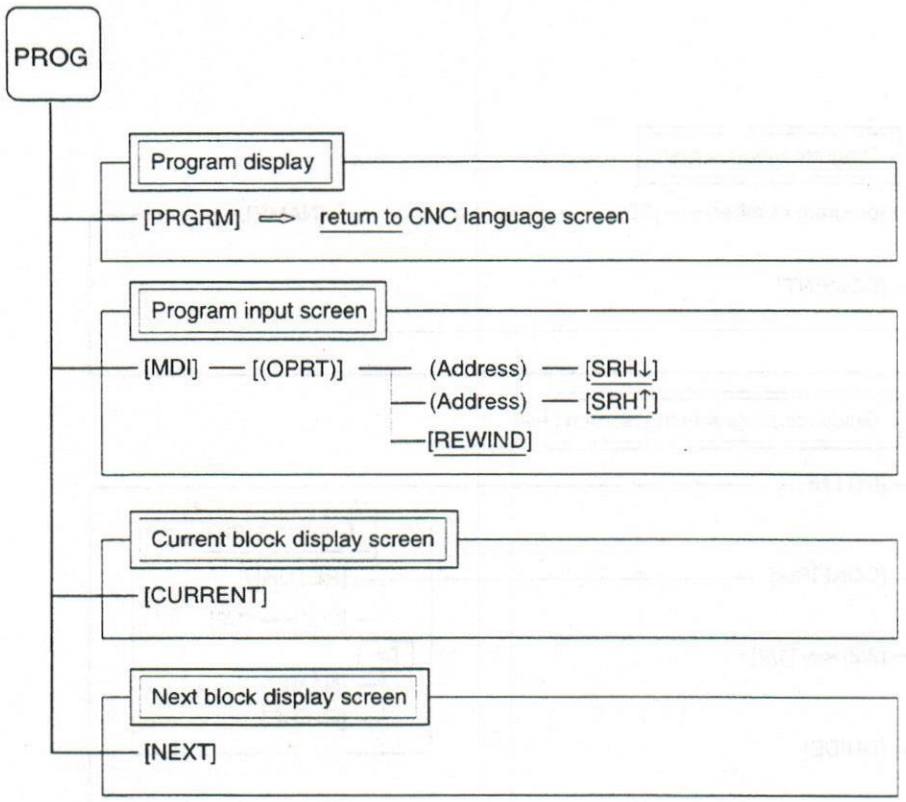


PROGRAM SCREEN Soft key transition triggered by the function key PROG in the MDI mode

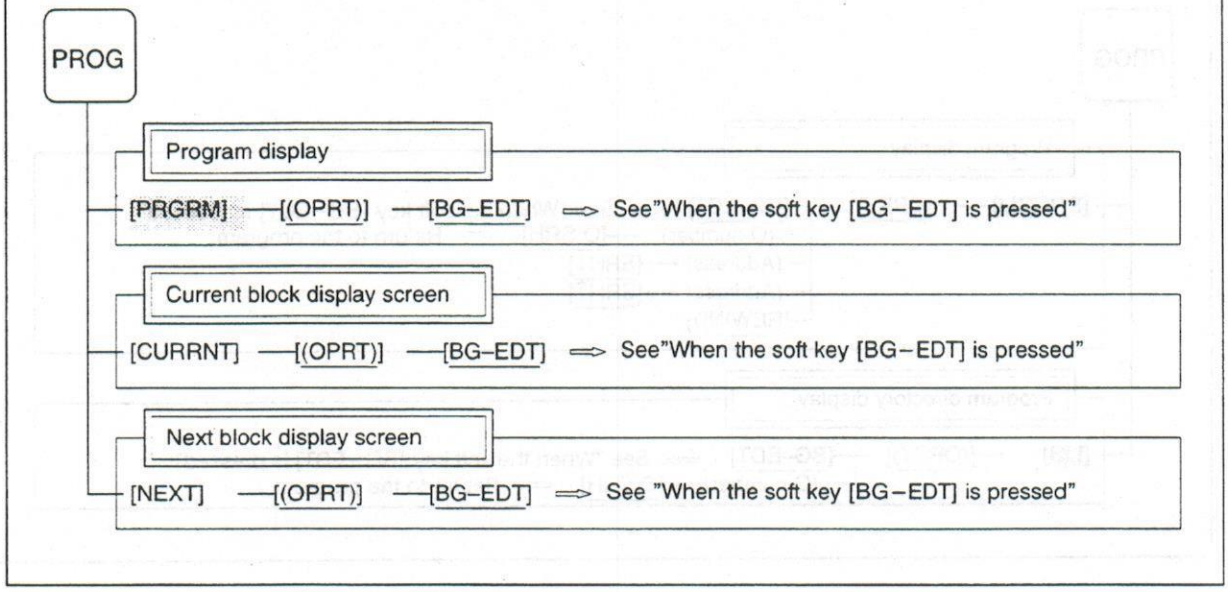


PROGRAM SCREEN Soft key transition triggered by the function key PROG in the MDI mode

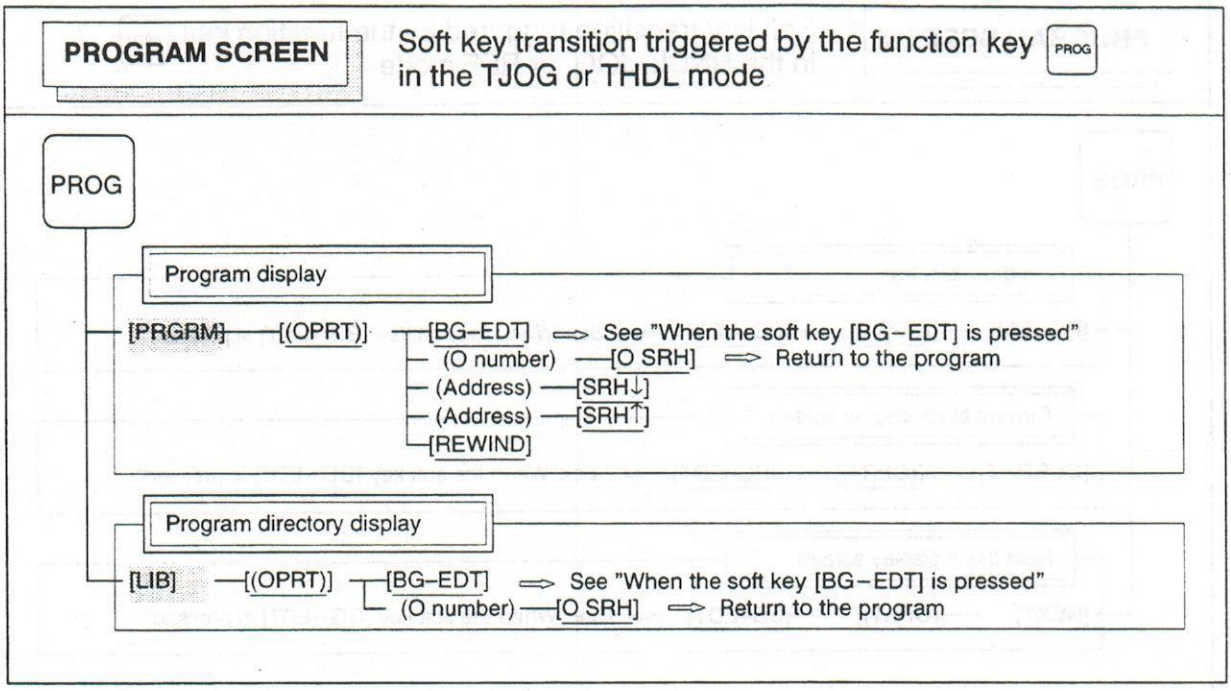
2/2



PROGRAM SCREEN Soft key transition triggered by the function key PROG in the HNDL, JOG, or REF mode

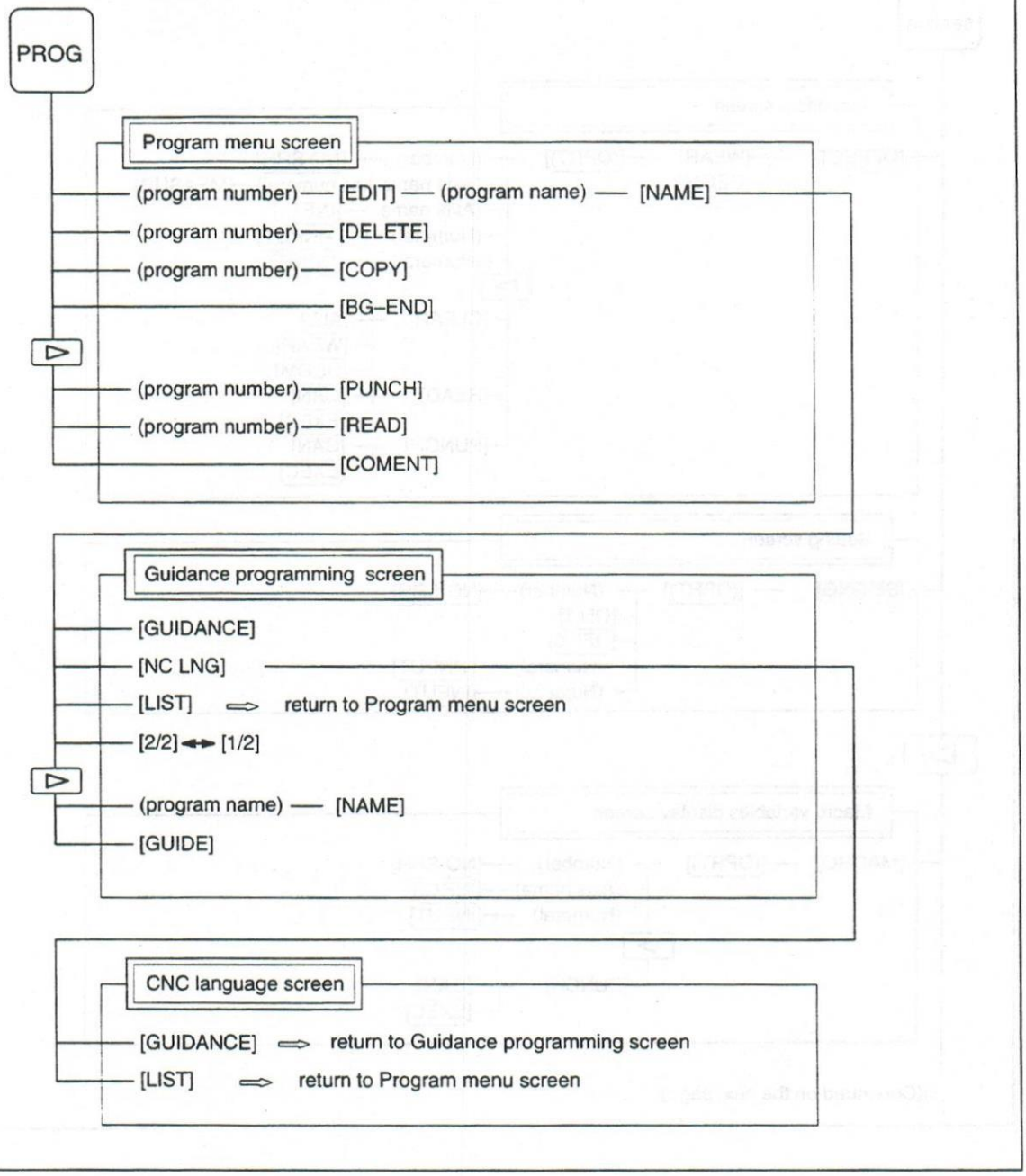

PROG
Program display
[PRGRM] —[(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
Current block display screen
[CURRNT] —[(OPRT)] —[BG–EDT] ⇒ See"When the soft key [BG–EDT] is pressed"
Next block display screen
[NEXT] —[(OPRT)] —[BG–EDT] ⇒ See "When the soft key [BG–EDT] is pressed"

PROGRAM SCREEN Soft key transition triggered by the function key PROG in the TJOG or THDL mode



PROGRAM SCREEN

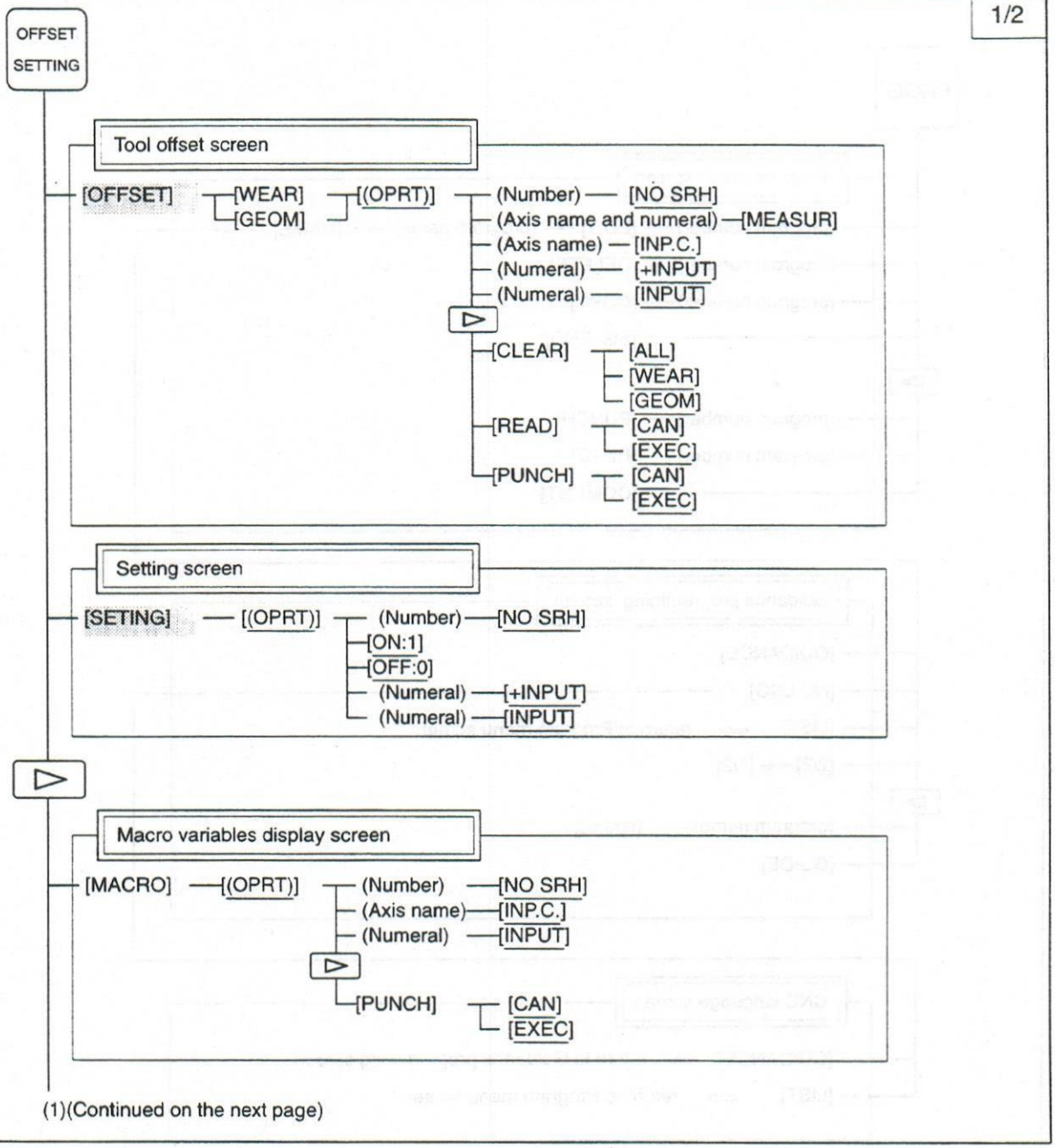
Soft key transition triggered by the function key PROG
(When the soft key [BG–EDT] is pressed in all modes)



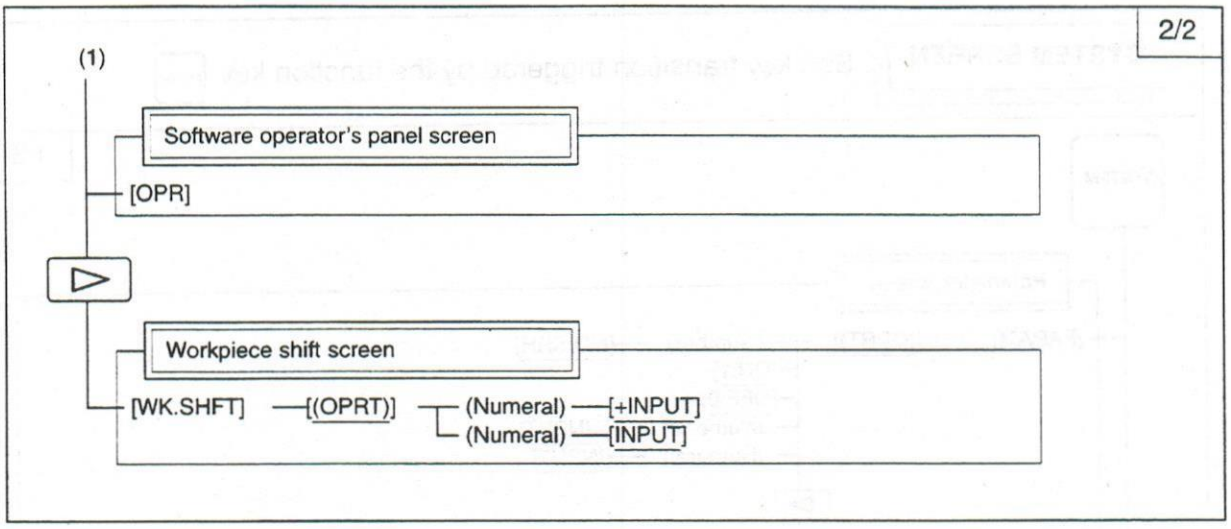
OFFSET/SETTING SCREEN

Soft key transition triggered by the function key OFFSET SETTING

1/2

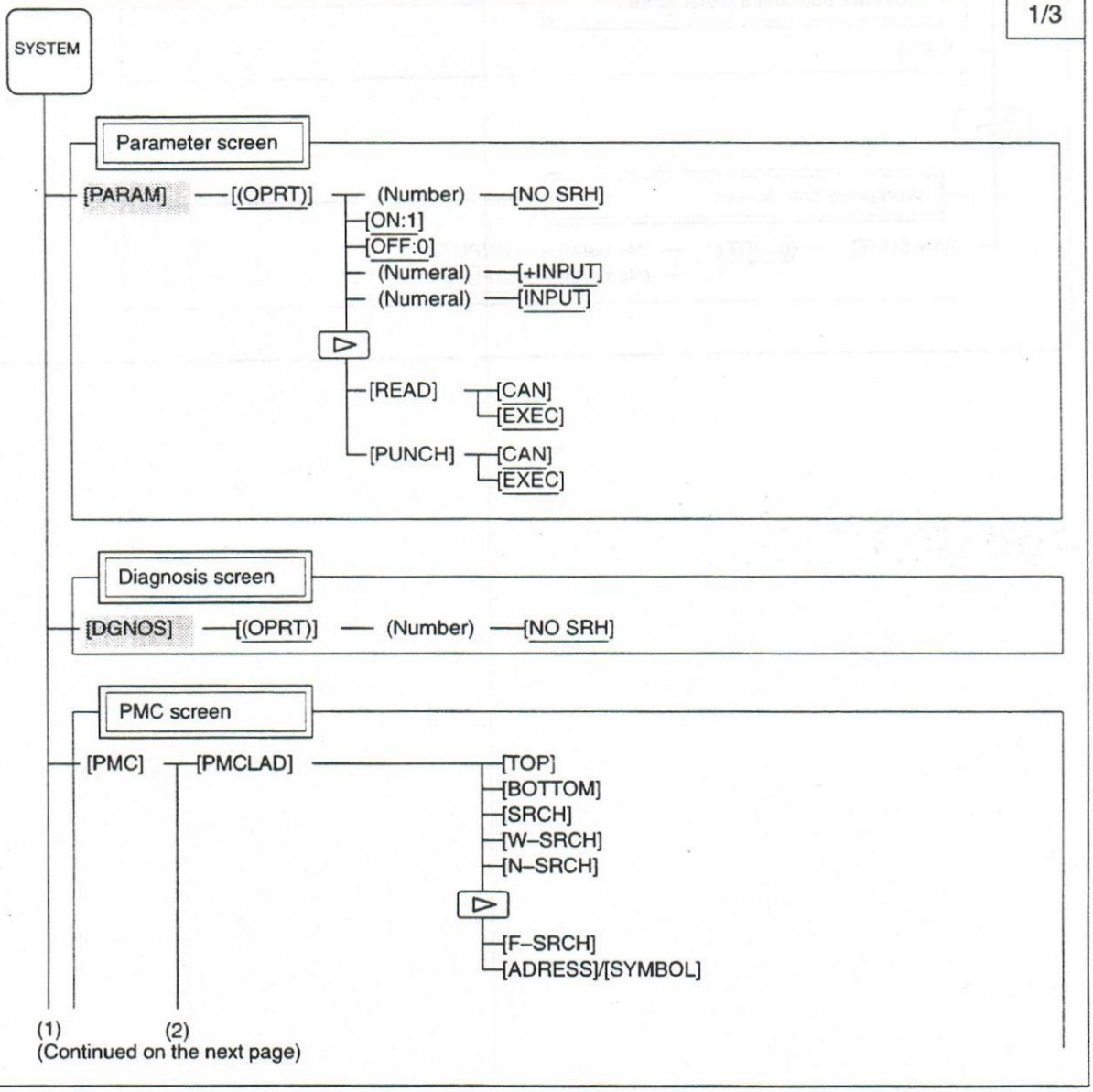


(1)(Continued on the next page)

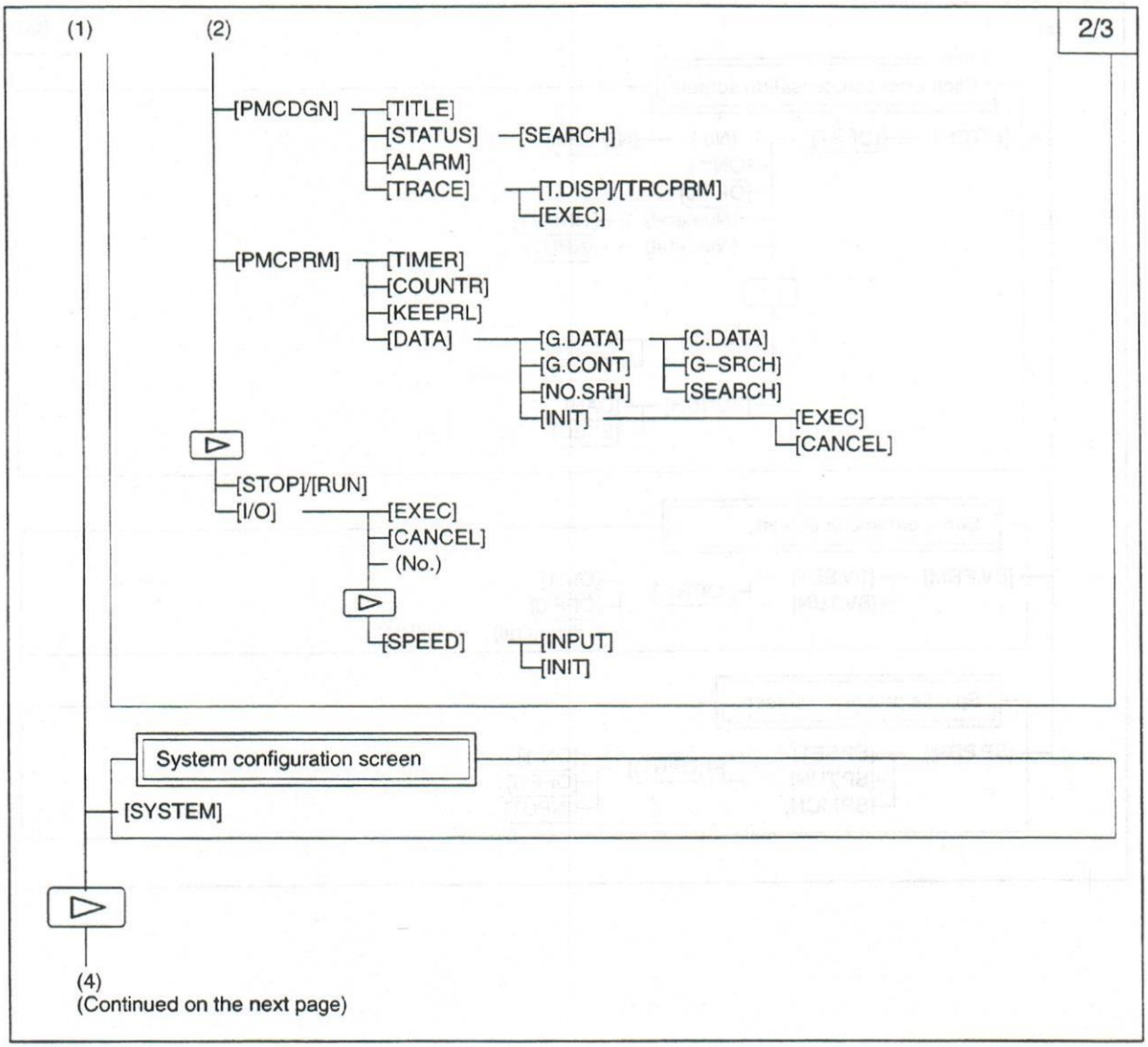

2/2
(1)
Software operator's panel screen
[OPR]
Workpiece shift screen
[WK.SHFT]
{(OPRT)}
(Numeral)
[+INPUT]
(Numeral)
[INPUT]

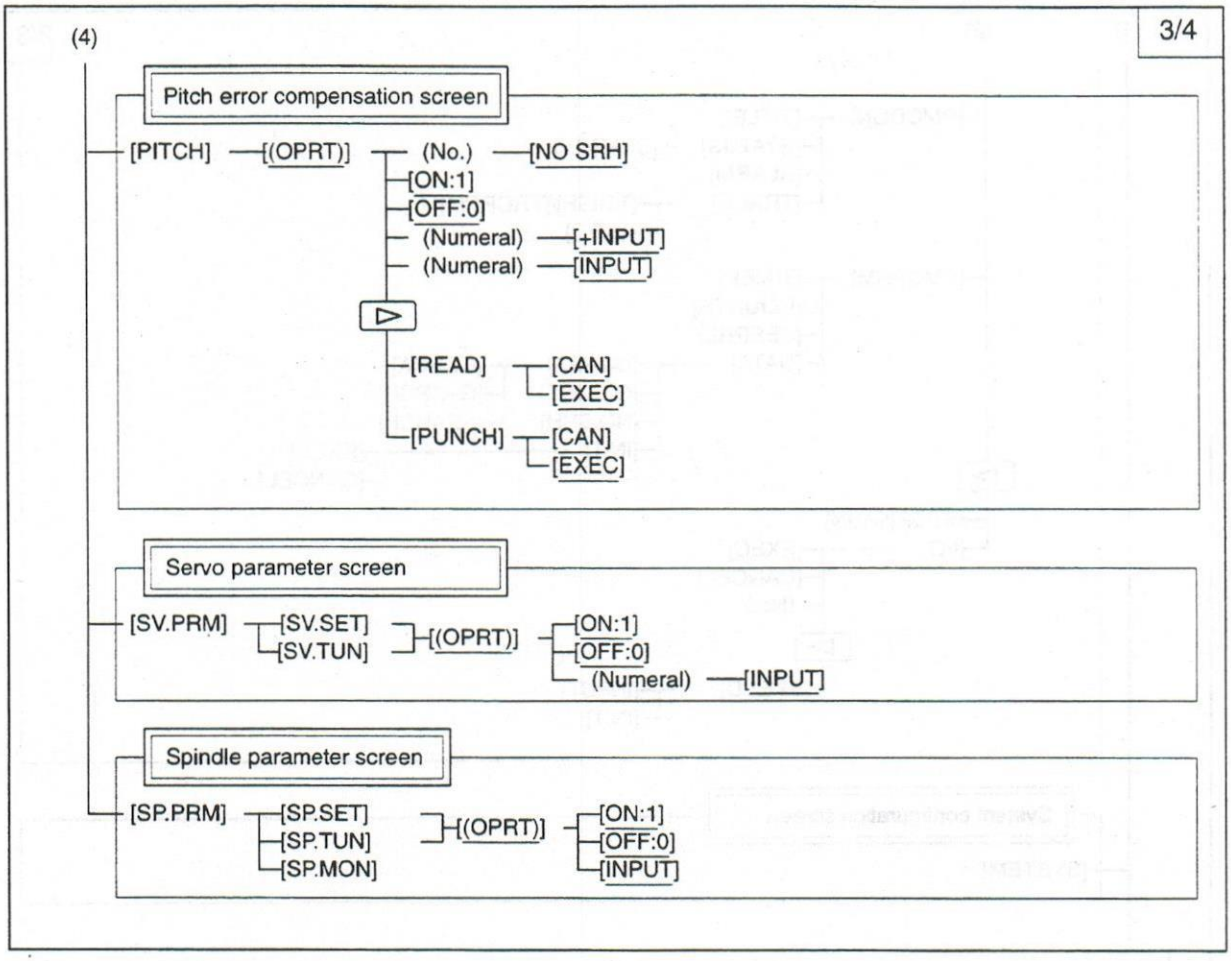
SYSTEM SCREEN Soft key transition triggered by the function key SYSTEM

1/3

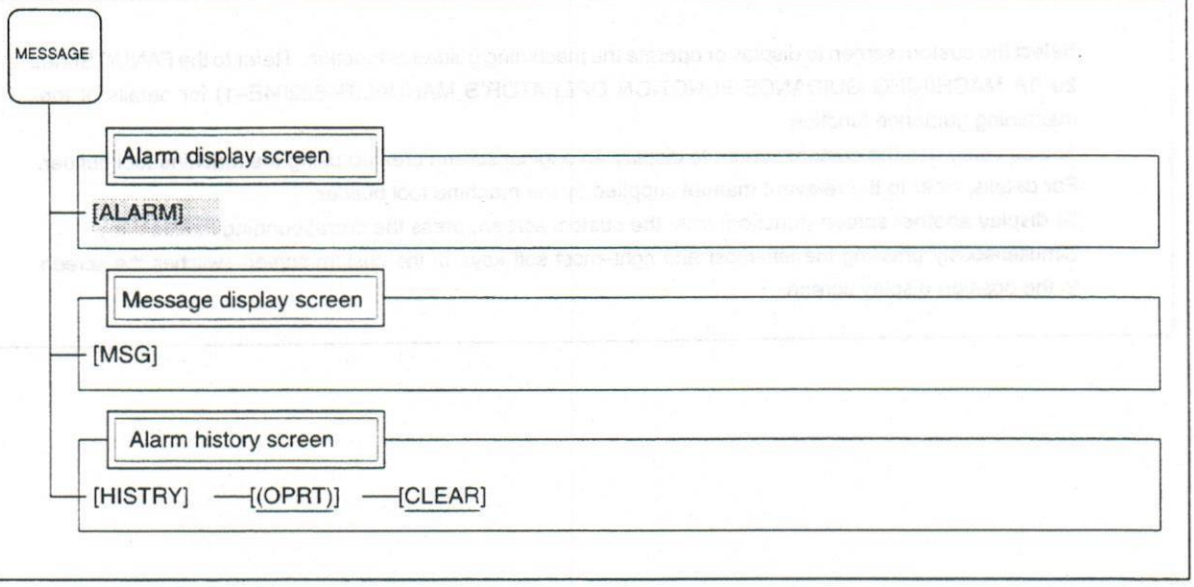


(Continued on the next page)

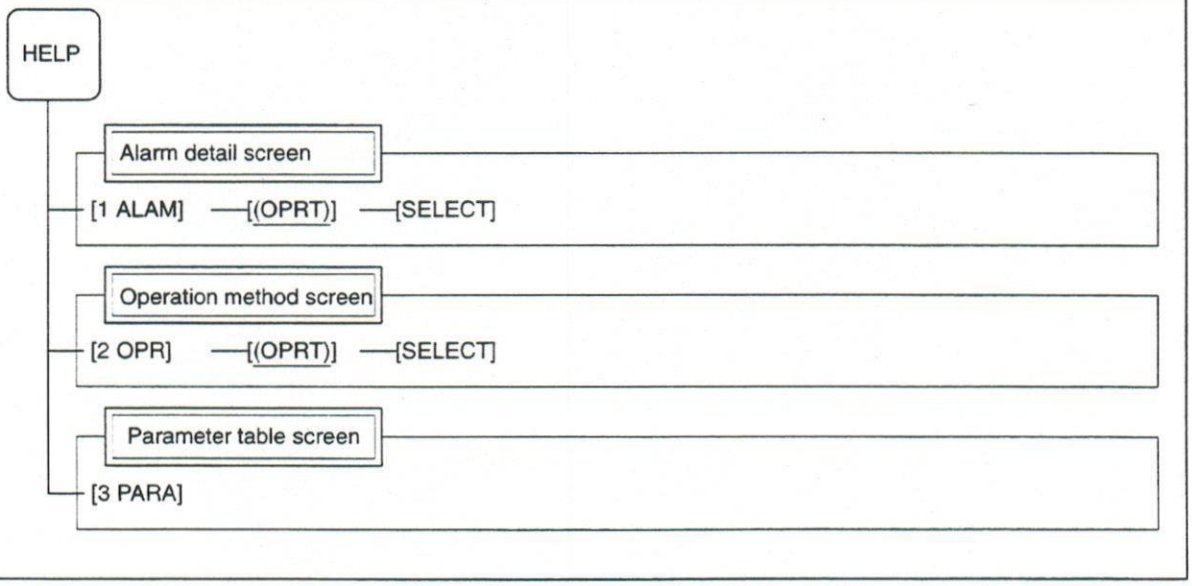

(1)
(2)
2/3
[PMCDGN]
[TITLE]
[STATUS]
[SEARCH]
[ALARM]
[TRACE]
[T.DISP]/[TRCPRM]
[EXEC]
[PMCPRM]
[TIMER]
[COUNTR]
[KEEPRL]
[DATA]
[G.DATA]
[C.DATA]
[G.CONT]
[G–SRCH]
[NO.SRH]
[SEARCH]
[INIT]
[EXEC]
[CANCEL]
[STOP]/[RUN]
[I/O]
[EXEC]
[CANCEL]
(No.)
[SPEED]
[INPUT]
[INIT]
System configuration screen
[SYSTEM]
(4)
(Continued on the next page)


(4)
3/4
Pitch error compensation screen
[PITCH]
[(OPRT)]
(No.)
[NO SRH]
[ON:1]
[OFF:0]
(Numeral)
[+INPUT]
(Numeral)
[INPUT]
[READ]
[CAN]
[EXEC]
[PUNCH]
[CAN]
[EXEC]
Servo parameter screen
[SV.PRM]
[SV.SET]
[SV.TUN]
[(OPRT)]
[ON:1]
[OFF:0]
(Numeral)
[INPUT]
Spindle parameter screen
[SP.PRM]
[SP.SET]
[SP.TUN]
[SP.MON]
[(OPRT)]
[ON:1]
[OFF:0]
[INPUT]

MESSAGE SCREEN Soft key transition triggered by the function key MESSAGE



HELP SCREEN Soft key transition triggered by the function key HELP



CUSTOM SCREEN

Soft key transition triggered by the function key CUSTOM

Select the custom screen to display or operate the machining guidance function. Refer to the FANUC Series 20–TA MACHINING GUIDANCE FUNCTION OPERATOR'S MANUAL(B–62204E–1) for details of the machining guidance function.

You can also use the custom screen to display an original screen created using the machine tool builder. For details, refer to the relevant manual supplied by the machine tool builder.

To display another screen (function) from the custom screen, press the corresponding function key.

Simultaneously pressing the left–most and right–most soft keys of the custom screen switches the screen to the position display screen.

2.2.4 Key Input and Input Buffer

When an address and a numerical key are pressed, the character corresponding to that key is input once into the key input buffer. The contents of the key input buffer is displayed at the bottom of the CRT screen.

In order to indicate that it is key input data, a ">" symbol is displayed immediately in front of it. A "_" is displayed at the end of the key input data indicating the input position of the next character.

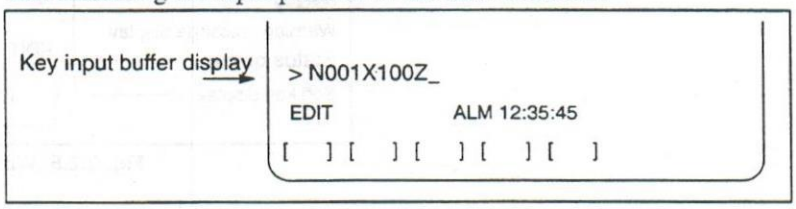


Fig. 2.2.4 Key input buffer display

To input the lower character of the keys that have two characters inscribed on them, first press the SHIFT key and then the key in question.

When the SHIFT key is pressed, "_" indicating the next character input position changes to " ∧ ". Now lowercase characters can be entered (shift state).

When a character is input in shift status the shift status is canceled. Furthermore, if the SHIFT key is pressed in shift status, the shift status is canceled.

It is possible to input up to 32 characters at a time in the key input buffer.

Press the CAN key to cancel a character or symbol input in the key input buffer.

(Example)

When the key input buffer displays
>N001X100Z_

and the cancel CAN key is pressed, Z is canceled and

>N001X100_
is displayed.

2.2.5 Warning Messages

After a character or number has been input from the MDI panel, a data check is executed when INPUT key or a soft key is pressed. In the case of incorrect input data or the wrong operation a flashing warning message will be displayed on the status display line.

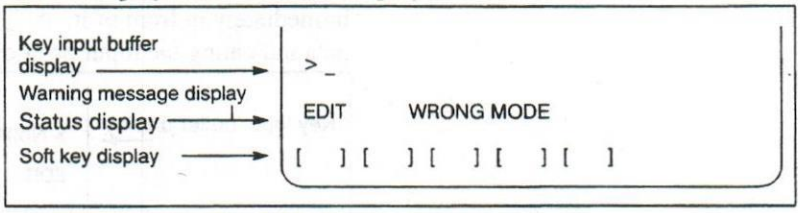


Fig. 2.2.5 Warning message display

Table2.2.5 Warning Messages

Warning message	Content
FORMAT ERROR	The format is incorrect.
WRITE PROTECT	Key input is invalid because of data protect key or the parameter is not write enabled.
DATA IS OUT OF RANGE	The input value exceeds the permitted range.
TOO MANY DIGITS	The input value exceeds the permitted number of digits.
WRONG MODE	Parameter input is not possible in any mode other than MDI mode.
EDIT REJECTED	It is not possible to edit in the current CNC status.

2.2.6 Function Selection Soft Keys

The function selection soft keys are used to select screens, in exactly the same way as function keys. For example, pressing the **[POS]** function selection soft key has the same effect as pressing the POS function key.

Function selection soft keys are displayed when bit 0 (FSK) of parameter No.3101 is set to 0. They are not displayed when the bit is set to 1.
The following function selection soft keys are supported :

```
[ POS ][ PRGRM ][SETTING][SYSTEM][ (CHAP) ]
```

```
[ MSG ][     ][CUSTOM][     ][ (CHAP) ]
```

The table below lists the correspondence between function selection soft keys and function keys.

Function selection soft key	Corresponding function key
[POS]	POS
[PRGRM]	PROG
[SETTING]	OFFSET SETTING
[SYSTEM]	SYSTEM
[MSG]	MESSAGE
[CUSTOM]	CUSTOM GRAPH

Pressing the **[(CHAP)]** soft key displays the chapter selection soft keys.

Note
This manual describes how to select screen by using function keys. The selection procedure is the same regardless of whether you use the function keys or function selection soft keys.

2.3 EXTERNAL I/O DEVICES

Five types of external input/output devices are available. This section outlines each device. For details on these devices, refer to the corresponding manuals listed below.

Table 2.3(a) External I/O device

Device name	Usage	Max. storage capacity	Reference manual
FANUC Handy File	Easy–to–use, multi function input/output device. It is designed for FA equipment and uses floppy disks.	3600m	B–61834 E
FANUC Floppy Cassette	Input/output device. Uses floppy disks.	2500m	B–66040 E
FANUC FA Card	Compact input/output device. Uses FA cards.	160m	B–61274 E
FANUC PPR	Input/output device consisting of a paper tape reader, tape punch, and printer.	275m	B–58584 E
Portable Tape Reader	Input device for reading paper tape.	———	Appendix H

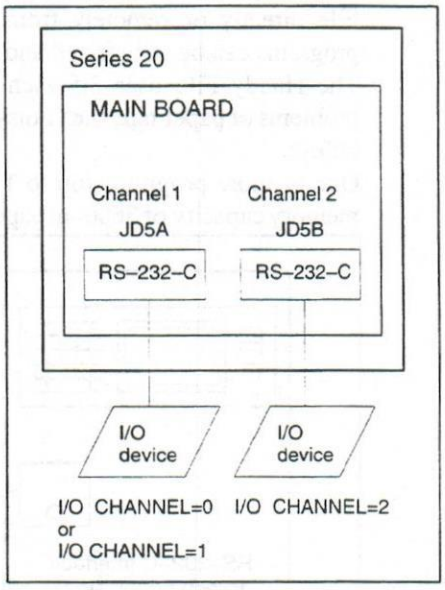
The following data can be input/output to or from external input/output devices:

1. **Programs**
2. **Offset data**
3. **Parameters**
4. **Custom macro common variables**
5. **Pitch error compensation data**

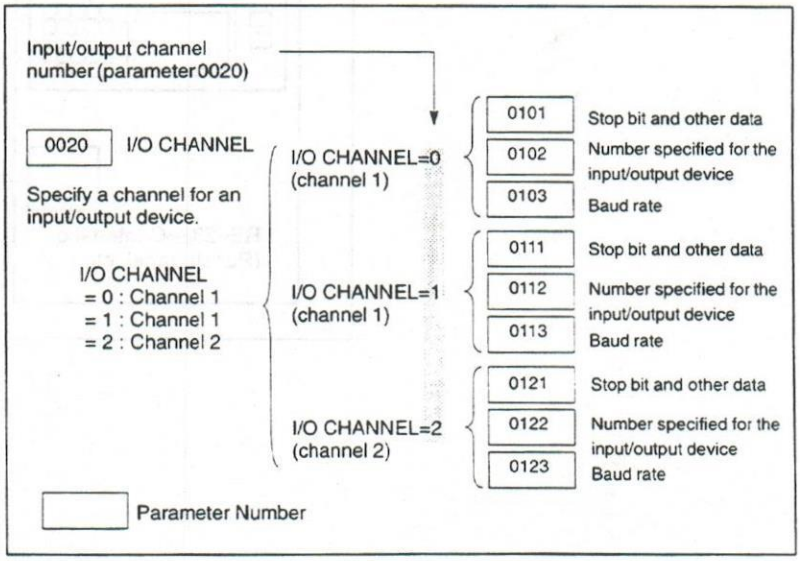
For how data is input and output, see Chapter 8.

Parameter

Before an external input/output device can be used, parameters must be set as follows.

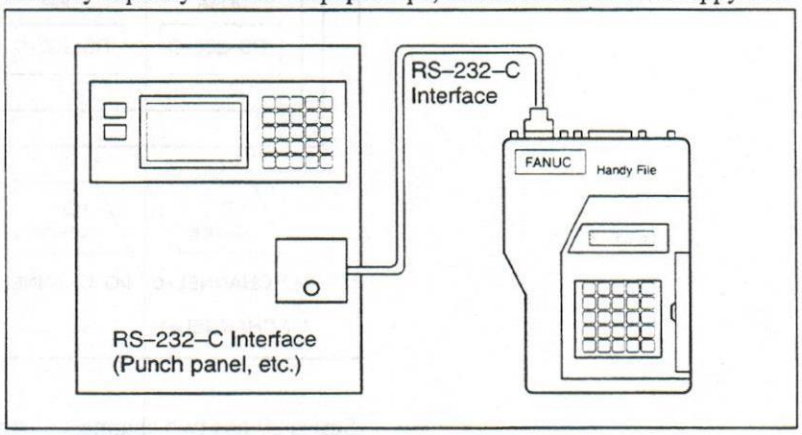


Series 20 has two channels of reader/punch interfaces. The input/output device to be used is specified by setting the channel connected to that device in setting data I/O CHANNEL.
The specified data, such as a baud rate and the number of stop bits, of an input/output device connected to a specific channel must be set in parameters for that channel in advance.
For channel 1, two combinations of parameters to specify the input/output device data are provided.
The following shows the interrelation between the reader/punch interface parameters for the channels.


Input/output channel number (parameter 0020)
0020 I/O CHANNEL
Specify a channel for an input/output device.
I/O CHANNEL
= 0 : Channel 1
= 1 : Channel 1
= 2 : Channel 2
I/O CHANNEL=0 (channel 1)
I/O CHANNEL=1 (channel 1)
I/O CHANNEL=2 (channel 2)
0101 Stop bit and other data
0102 Number specified for the input/output device
0103 Baud rate
0111 Stop bit and other data
0112 Number specified for the input/output device
0113 Baud rate
0121 Stop bit and other data
0122 Number specified for the input/output device
0123 Baud rate
Parameter Number

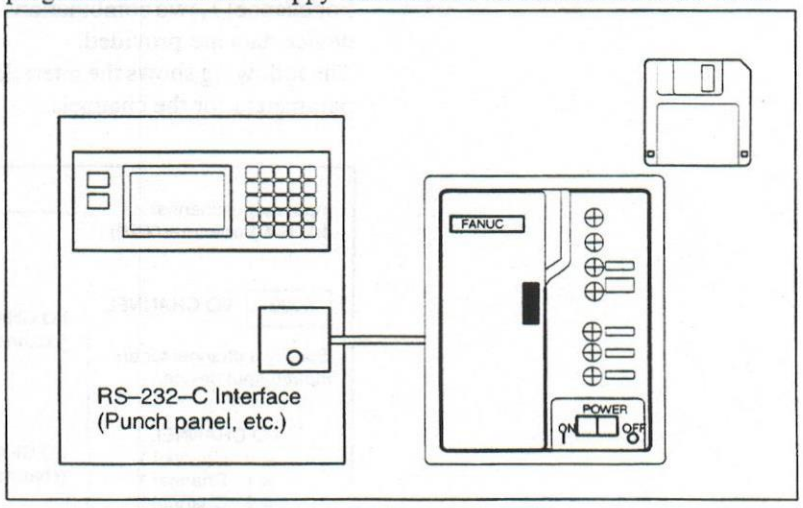
2.3.1 FANUC Handy File

The Handy File is an easy–to–use, multi function floppy disk input/output device designed for FA equipment. By operating the Handy File directly or remotely from a unit connected to the Handy File, programs can be transferred and edited.
The Handy File uses 3.5–inch floppy disks, which do not have the problems of paper tape (i.e., noisy during input/output, easily broken, and bulky).
One or more programs (up to 1.44M bytes, which is equivalent to the memory capacity of 3600–m paper tape) can be stored on one floppy disk.



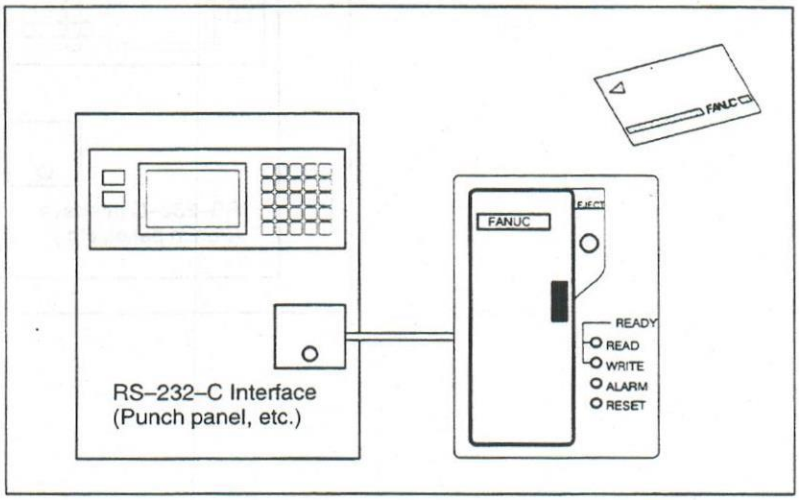
2.3.2 FANUC Floppy Cassette

When the Floppy Cassette is connected to the CNC, machining programs stored in the CNC can be saved on a Floppy Cassette, and machining programs saved in the Floppy Cassette can be transferred to the CNC.



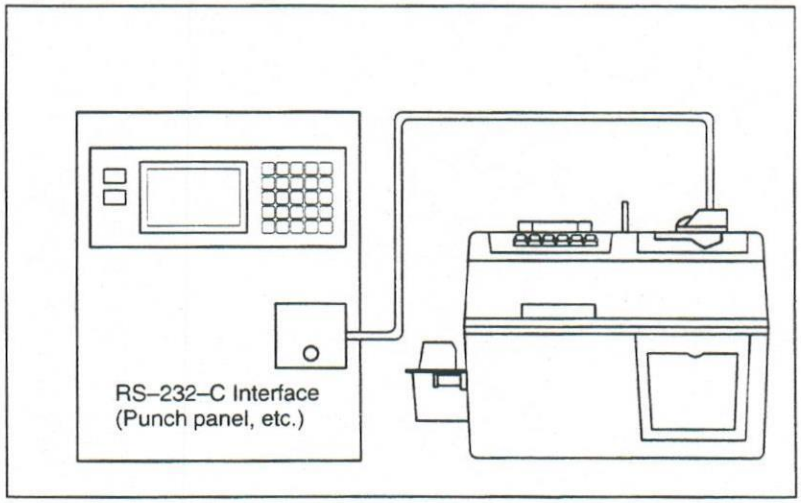
2.3.3 FANUC FA Card

An FA Card is a memory card used as an input medium in the FA field. It is compact, but has a large memory capacity with high reliability, and requires no special maintenance.
When an FA Card is connected to the CNC via the card adapter, machining programs stored in the CNC can be transferred to and saved in an FA Card. Machining programs stored on an FA Card can also be transferred to the CNC.



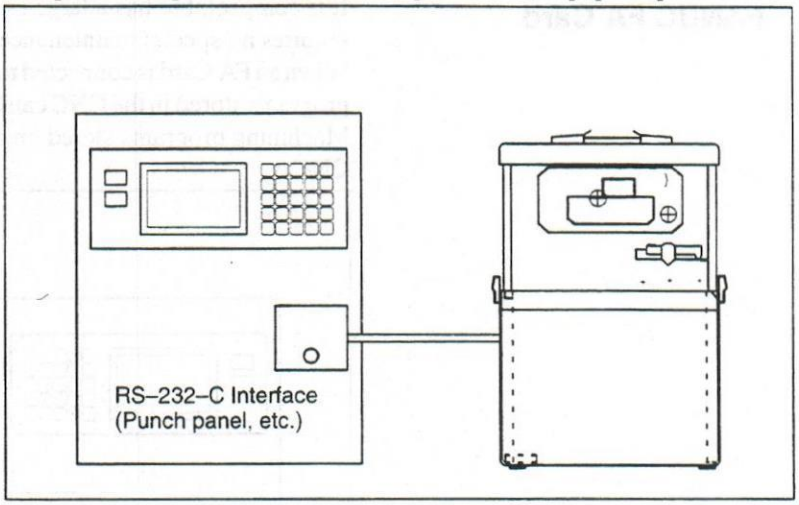
2.3.4 FANUC PPR

The FANUC PPR consists of three units: A printer, paper tape punch, and paper tape reader.
When the PPR is used alone, data can be read from the tape reader and printed or punched out. It is also possible to perform TH and TV checks on data that was read.



2.3.5 Portable Tape Reader

The portable tape reader is used to input data from paper tape.



2.4 POWER ON/OFF

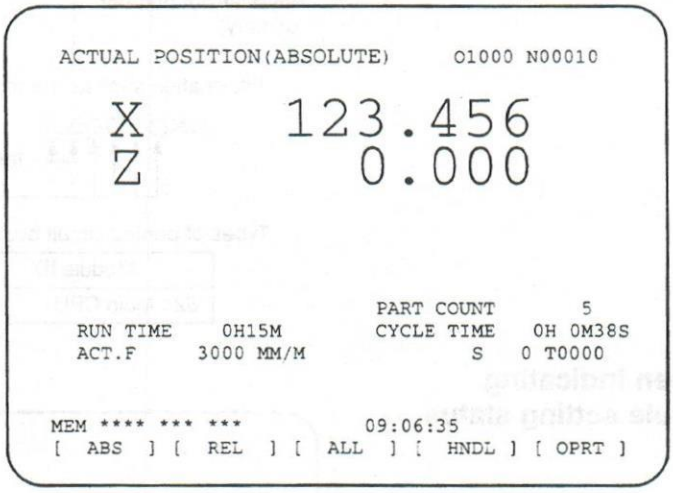
2.4.1 Turning on the Power

Procedure of turning on the power

1 Check that the appearance of the CNC machine tool is normal. (For example, check that front door and rear door are closed.)

2 Turn on the power according to the manual issued by the machine tool builder.

3 After the power is turned on, check that the position screen is displayed. An alarm has occured at power–on, however, the corresponding alarm screen is displayed. If the screen shown in Section 2.4.2 is displayed, a system failure may have occurred.


ACTUAL POSITION(ABSOLUTE) O1000 N00010
X 123.456
Z 0.000
PART COUNT 5
RUN TIME 0H15M CYCLE TIME 0H 0M38S
ACT.F 3000 MM/M S 0 T0000
MEM **** *** ***
09:06:35
[ABS] [REL] [ALL] [HNDL] [OPRT]

4 Check that the fan motor is rotating.

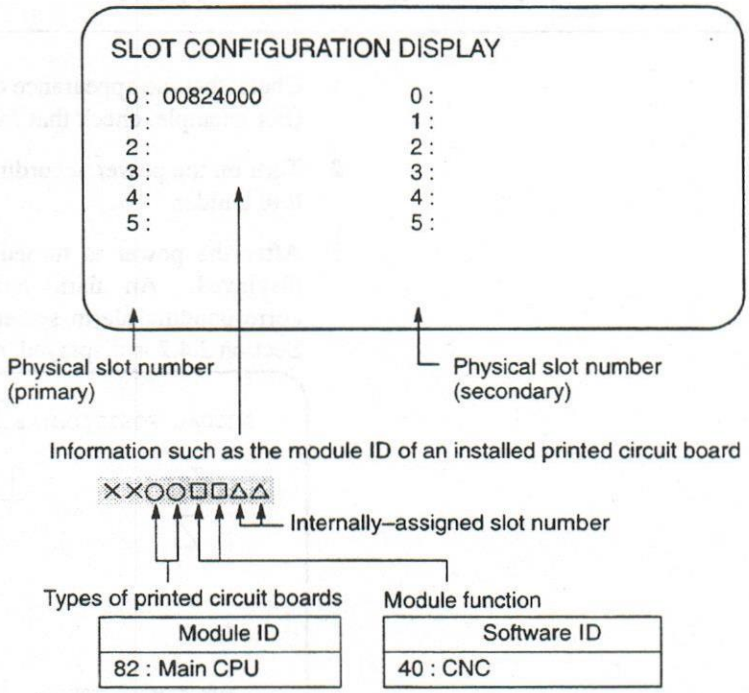
Note

When the power is turned on, do not touch CRT/MDI panel keys until the positional or alarm screen is displayed. Some keys are used for the maintenance or special operation purpose. When they are pressed, unexpected operation may be caused.

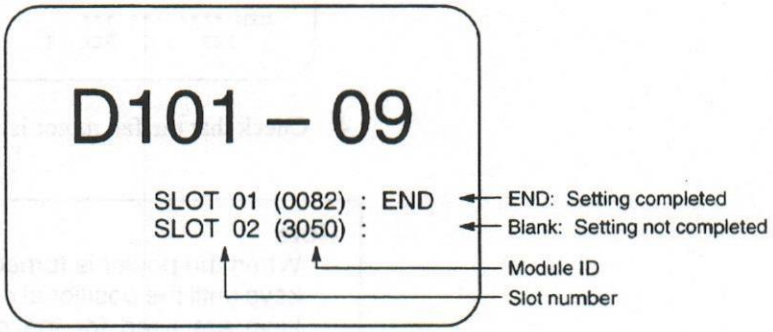
2.4.2 Screen Displayed at Power–on

If a hardware failure or installation error occurs, the system displays one of the following three types of screens then stops.
Information such as the type of printed circuit board installed in each slot is indicated. This information and the LED states are useful for failure recovery.

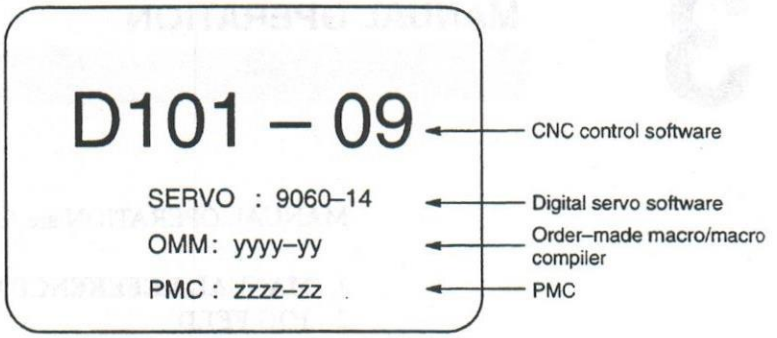
Slot status display



Screen indicating module setting status



Display of software configuration



The software configuration can also be displayed on the system configuration screen.

2.4.3 Power Disconnection

Procedure for Poser Disconnection

1 Check that the LED indicating the cycle start is off on the operator's panel.

2 Check that all movable parts of the CNC machine tool is stopping.

3 If an external input/output device such as the Handy File is connected to the CNC, turn off the external input/output device.

4 Continue to press the POWER OFF pushbutton for about 5 seconds.

5 Refer to the machine tool builder's manual for turning off the power to the machine.

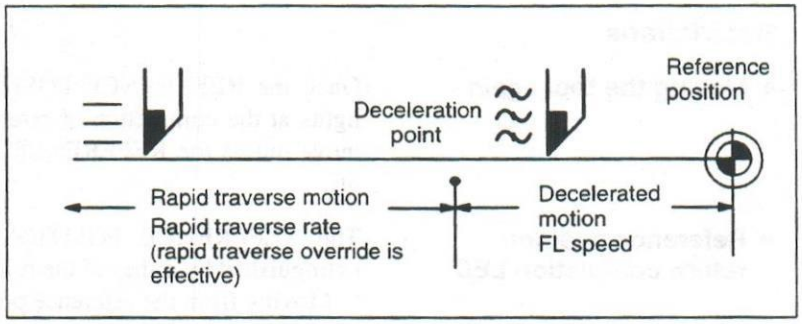
3 MANUAL OPERATION

MANUAL OPERATION are four kinds as follows :

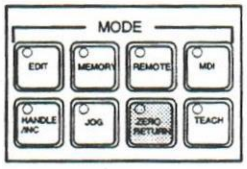
1. **MANUAL REFERENCE POSITION RETURN**
2. **JOG FEED**
3. **INCREMENTAL FEED**
4. **MANUAL HANDLE FEED**
5. **MANUAL ABSOLUTE ON/OFF**

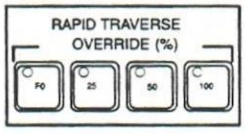
3.1 MANUAL REFERENCE POSITION RETURN

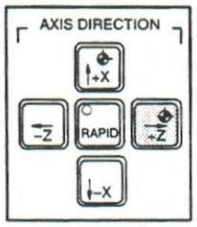
The tool is returned to the reference position as follows :
The tool is moved in the direction specified in parameter ZMI (bit 5 of No. 1006) for each axis with the reference position return switch on the machine operator's panel. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed. The rapid traverse rate and FL speed are specified in parameters (No. 1424, 1421, and 1425).
Four step rapid traverse override is effective during rapid traverse.
When the tool has returned to the reference position, the reference position return completion LED goes on. The tool generally moves along only a single axis, but can move along two axes simultaneously when specified so in parameter JAX(bit 0 of No.1002).



Procedure for Manual Reference Position Return





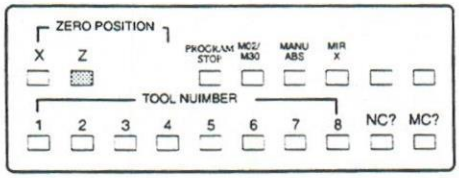


1 Press the reference position return switch, one of the mode selection switches.

2 To decrease the feedrate, press a rapid traverse override switch.

3 Press the feed axis and direction selection switch corresponding to the axis and direction for reference position return. Continue pressing the switch until the tool returns to the reference position. The tool can be moved along two axes simultaneously when specified so in an appropriate parameter setting. The tool moves to the deceleration point at the rapid traverse rate, then moves to the reference position at the FL speed set in a parameter.
When the tool has returned to the reference position, the reference position return completion LED goes on.

4 Perform the same operations for other axes, if necessary.
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.



Explanation

- **Automatically setting the coordinate system**

Bit 0 (ZPR) of parameter No. 1201 is used for automatically setting the coordinate system. When ZPR is set, the coordinate system is automatically determined when manual reference position return is performed.
When α and γ are set in parameter 1250, the workpiece coordinate system is determined so that the reference point on the tool holder or the position of the tip of the reference tool is X=α,Z=γwhen reference position return is performed. This has the same effect as specifying the following command for reference position return:
G50Xα_Zγ_;

Restrictions

- **Moving the tool again**

Once the REFERENCE POSITION RETURN COMPLETION LED lights at the completion of reference position return, the tool does not move unless the REFERENCE POSITION RETURN switch is turned off.

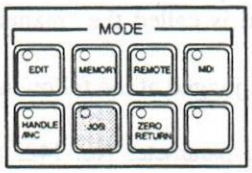
- **Reference position return completion LED**

The REFERENCE POSITION RETURN COMPLETION LED is extinguished by either of the following operations:
– Moving from the reference position.
– Entering an emergency stop state.

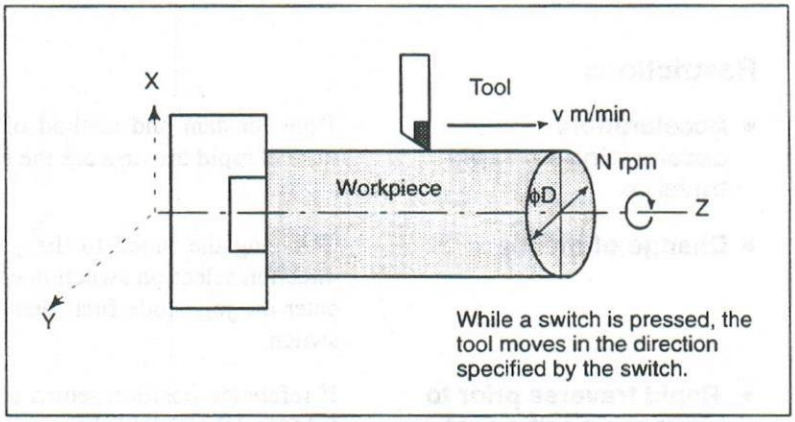
- **The distance to return to reference position**

For the distance (Not in the deceleration condition) to return the tool to the reference position, refer to the manual issued by the machine tool builder.

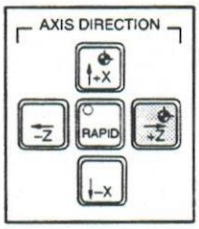
3.2 JOG FEED

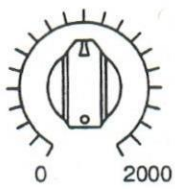


In the jog mode, pressing a feed axis and direction selection switch on the machine operator's panel continuously moves the tool along the selected axis in the selected direction.
The jog feedrate is specified in a parameter (No.1423)
The jog feedrate can be adjusted with the jog feedrate override dial.
Pressing the rapid traverse switch moves the tool at the rapid traverse feedrate (parameter No.1424) regardless of the position of the jog feedrate override dial.
Manual operation is allowed for one axis at a time. 2 axes can be selected at a time by parameter JAX (No.1002#0).

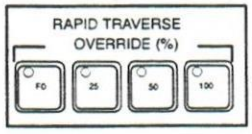


Procedure for Jog Feed





JOG FEED RATE OVERRIDE



1 Press the jog switch, one of the mode selection switches.

2 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. While the switch is pressed, the tool moves at the feedrate specified in a parameter (No. 1423). The tool stops when the switch is released.

3 The jog feedrate can be adjusted with the jog feedrate override dial.

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate while the rapid traverse switch is pressed. Rapid traverse override by the rapid traverse override switches is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanations

- **Manual synchronous feed**

By setting the parameter JRV (No. 1402#4) to 1, jog feed becomes a feed synchronized with spindle revolution. The feed is called the "manual synchonous feed".
During the manual synchronous feed, the tool is jogged at the following feedrate:
Feed amount per spindle revolution specified by parameter No. 1423 × spindle speed.
Also, the jog feedrate is applied to this rate.
Therefore, thr feedrate during the manual synchronous feed follows:
Value of parameter No. 1423 × spindle speed × jog feedrate override.

Restrictions

- **Acceleration/ deceleration for rapid traverse**

Time constant and method of automatic acceleration/ deceleration for manual rapid traverse are the same as G00 in programmed command.

- **Change of modes**

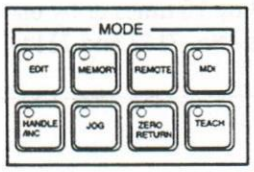
Changing the mode to the jog mode while pressing a feed axis and direction selection switch does not enable jog feed. To enable jog feed, enter the jog mode first, then press a feed axis and direction selection switch.

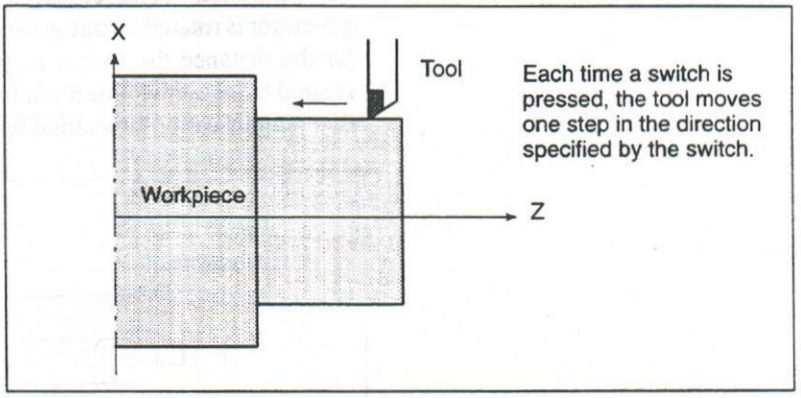
- **Rapid traverse prior to reference position return**

If reference position return is not performed after power–on, pushing RAPID TRAVERSE button does not actuate the rapid traverse but the remains at the JOG feedrate. This function can be disabled by setting parameter RPD (No.1401#01).

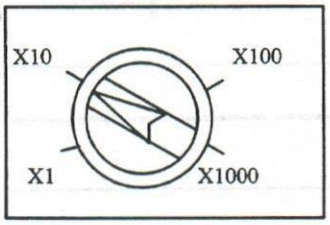
3.3 INCREMENTAL FEED

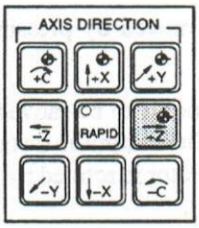
In the handle/incremental mode, pressing a feed axis and direction selection switch on the machine operator's panel moves the tool one step along the selected axis in the selected direction. The minimum distance the tool is moved is the least input increment. Each step can be 10, 100, or 1000 times the least input increment.





Procedure For Incremental Feed





1 Press the HANDLE/INC switch, one of the mode selection switches.

2 Select the distance to be moved for each step with the magnification dial.

3 Press the feed axis and direction selection switch corresponding to the axis and direction the tool is to be moved. Each time a switch is pressed, the tool moves one step. The feedrate is the same as the jog feedrate.

4 Pressing the rapid traverse switch while pressing a feed axis and direction selection switch moves the tool at the rapid traverse rate. Rapid traverse override by the rapid traverse override switch is effective during rapid traverse.

The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Explanation

- **Travel distance specified with a diameter**

The distance the tool travels along the X–axis can be specified with a diameter.

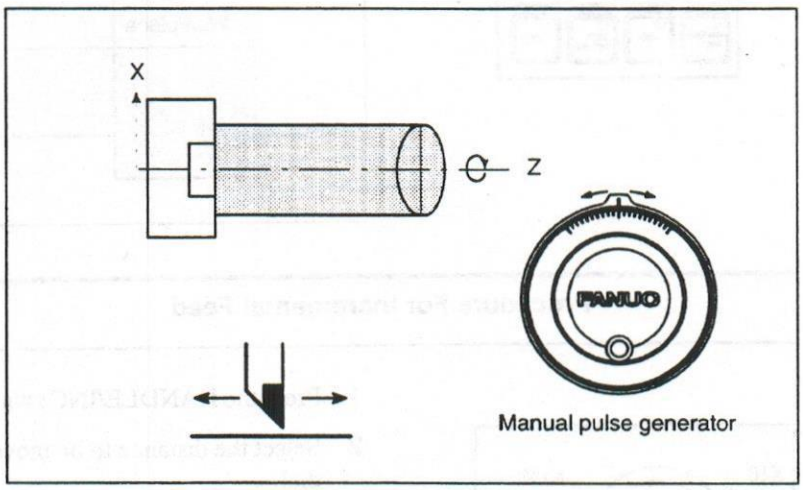
- **Handle/incremental feed mode**

When bit 0 (JHD) of parameter No. 7100 is set to 0, only manual handle feed is enabled in handle/incremental feed mode. When bit 0 (JHD) of parameter No. 7100 is set to 1, manual handle feed and incremental feed are enabled in handle/incremental feed mode. Manual handle feed and incremental feed, however, cannot be enabled at the same time. Manual handle feed is possible only when the tool is not being moved by incremental feed.

3.4 MANUAL HANDLE FEED

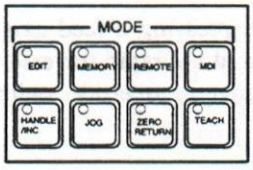
In the handle/incremeutal feed mode, the tool can be minutely moved by rotating the manual pulse generator on the machine operator's panel. Select the axis along which the tool is to be moved with the handle feed axis selection switches.

The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment. Or the distance the tool is moved when the manual pulse generator is rotated by one graduation can be magnified by 10 times or by one of the two magnifications specified by parameters (No. 7113 and 7114).



Manual pulse generator

Procedure for Manual Handle Feed



1 Press the HANDLE/INC switch, one of the mode selection switches.

2 Select the axis along which the tool is to be moved by pressing a handle feed axis selection switch.

3 Select the magnification for the distance the tool is to be moved by pressing a handle feed magnification switch. The minimum distance the tool is moved when the manual pulse generator is rotated by one graduation is equal to the least input increment.

4 Move the tool along the selected axis by rotating the handle. Rotating the handle 360 degrees moves the tool the distance equivalent to 100 graduations.
The above is an example. Refer to the appropriate manual provided by the machine tool builder for the actual operations.

Manual pulse generator

Explanation

- **Availability of manual pulse generator in Jog mode (JHD)**

Parameter JHD (bit 0 of No. 7100) enables or disables the manual handle feed in the JOG mode.
When the parameter JHD(bit 0 of No. 7100) is set 1,both manual handle feed and incremental feed are enabled.

- **Availability of manual pulse generator in TEACH IN JOG mode (THD)**

Parameter THD (bit 1 of No. 7100) enables or disables the manual handle feed in the TEACH IN JOG mode.

- **A command to the MPG exceeding rapid traverse rate**

Parameter HPF (No.7117) specifies as follows:

SET VALUE 0 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are ignored.(The distance the tool is moved may not match the graduations on the manual pulse generator.)

SET VALUE 1 : The feedrate is clamped at the rapid traverse rate and generated pulses exceeding the rapid traverse rate are not ignored but accumulated in the CNC until the limit specified in parameter No.7117 is reached.
(No longer rotating the handle does not immediately stop the tool. The tool is moved by the pulses accumulated in the CNC before it stops.)

- **Movement direction of an axis to the rotation of MPG (HNG)**

Parameter HNG (No. 7102#0) switches the direction in which the tool moves along an axis, corresponding to the direction in which the handle of the manual pulse generator is rotated.

Restrictions

- **Number of MPGs**

Up to three manual pulse generators can be connected.

Notes

1. Rotate the manual pulse generator at a rate of five rotations per second or lower. If the manual pulse generator is rotated at a rate higher than five rotations per second, the tool may not stop immediately after thehandle is no longer rotated or the distance the tool moves may not match the graduations on the manual pulse generator.
2. Rotating the handle quickly with a large magnification such as x100 moves the tool too fast. The feedrate is clamped at the rapid traverse feedrate.

3.5 MANUAL ABSOLUTE ON AND OFF

Whether the distance the tool is moved by manual operation is added to the coordinates can be selected by turning the manual absolute switch on or off on the machine operator's panel. When the switch is turned on, the distance the tool is moved by manual operation is added to the coordinates. When the switch is turned off, the distance the tool is moved by manual operation is not added to the coordinates.

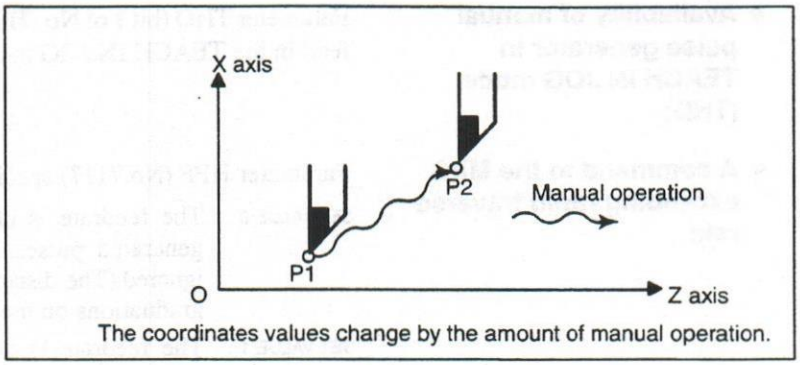


Fig. 3.5 (a) Coordinates with the switch ON

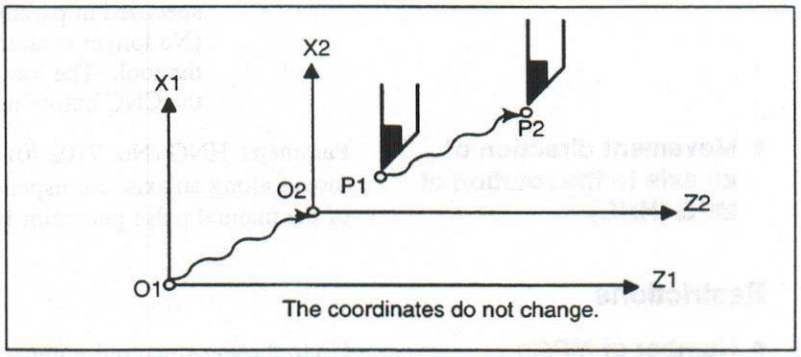


Fig. 3.5 (b) Coordinates with the switch OFF

Explanation

The following describes the relation between manual operation and coordinates when the manual absolute switch is turned on or off, using a program example.

```
G01    X100.0Z100.0F010 ;    (1)
       X150.0Z200.0     ;    (2)
       X200.0Z300.0     ;    (3)
```

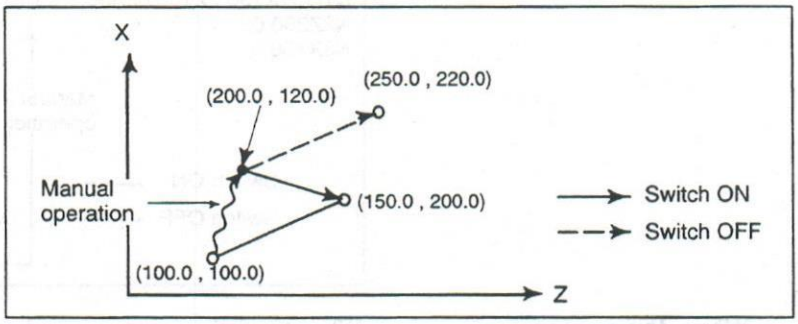
The subsequent figures use the following notation:

——→ Movement of the tool when the switch is on
– – → Movement of the tool when the switch is off

The coordinates after manual operation include the distance the tool is moved by the manual operation. When the switch is off, therefore, subtract the distance the tool is moved by the manual operation.

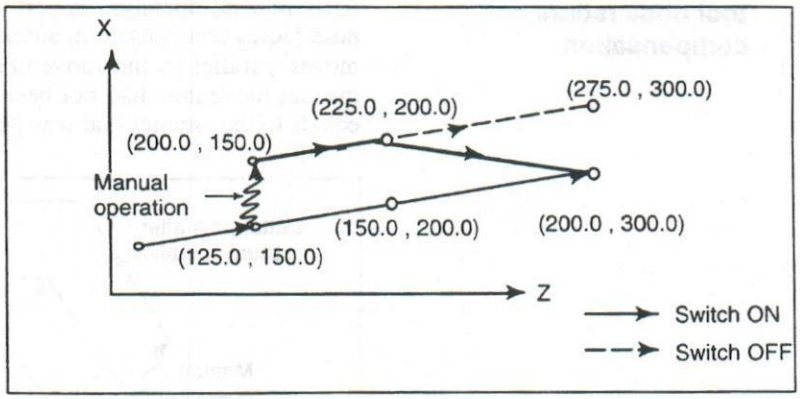
- **Manual operation after the end of block**

Coordinates when block (2) has been executed after manual operation (X–axis +100.0, Z–axis +20.0) at the end of movement of block (1).



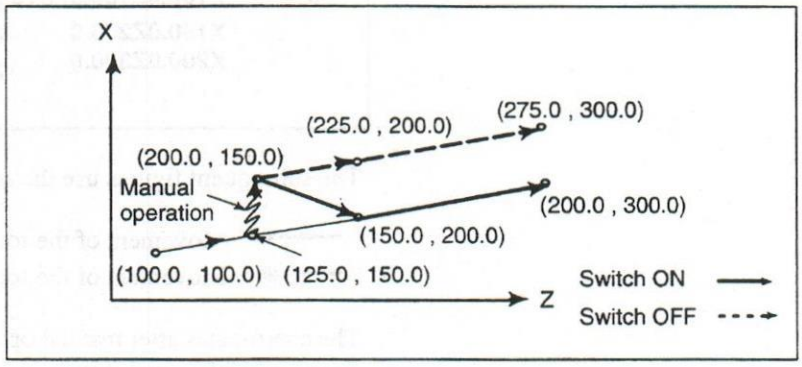
- **Manual operation after a feed hold**

Coordinates when the feed hold button is pressed while block (2) is being executed, manual operation (X–axis + 75.0) is performed, and the cycle start button is pressed and released



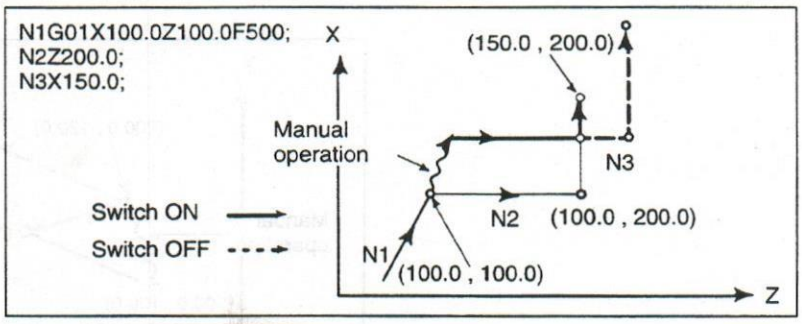
- **When reset after a manual operation following a feed hold**

Coordinates when the feed hold button is pressed while block (2) is being executed, manual operation (Y–axis +75.0) is performed, the control unit is reset with the RESET button, and block (2) is read again



- **When a movement command in the next block is only one axis**

When there is only one axis in the following command, only the commanded axis returns.



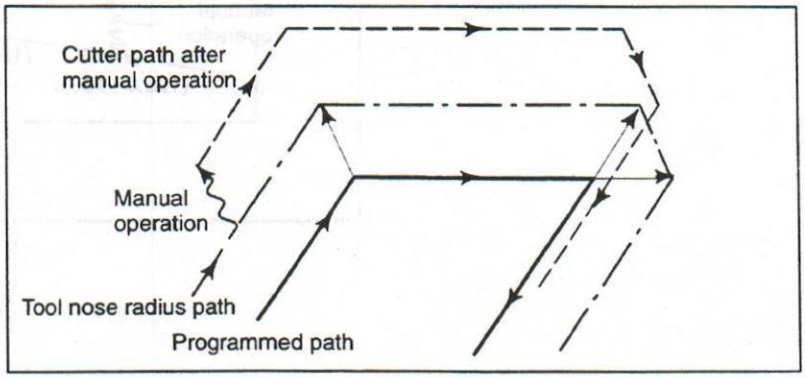
- **When the next move block is an incremental**

When the following commands are incremental commands, operation is the same as when the switch is OFF.

- **Manual operation during tool nose radius compensation**

When the switch is OFF

After manual operation is performed with the switch OFF during tool nose radius compensation, automatic operation is restarted then the tool moves parallel to the movement that would have been performed if manual movement had not been performed. The amount of separation equals to the amount that was performed manually.

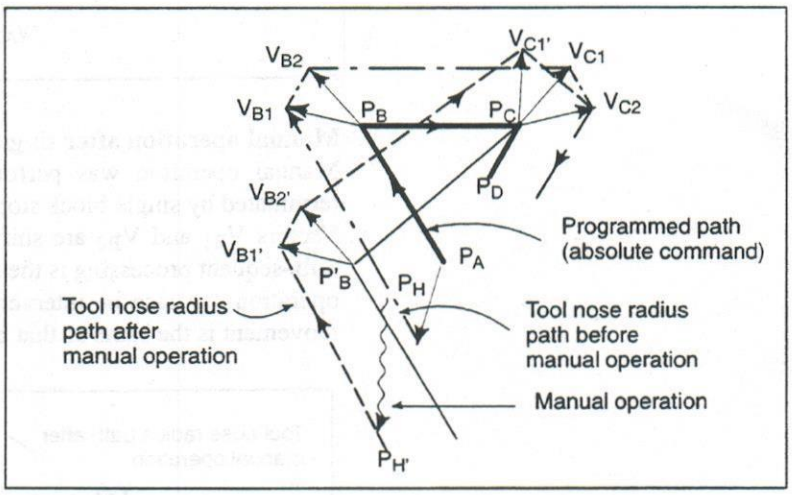


When the switch is ON during tool nose radius compensation

Operation of the machine upon return to automatic operation after manual intervention with the switch is ON during execution with an absolute command program in the tool nose radius compensation mode will be described. The vector created from the remaining part of the current block and the beginning of the next block is shifted in parallel. A new vector is created based on the next block, the block following the next block and the amount of manual movement. This also applies when manual operation is performed during cornering.

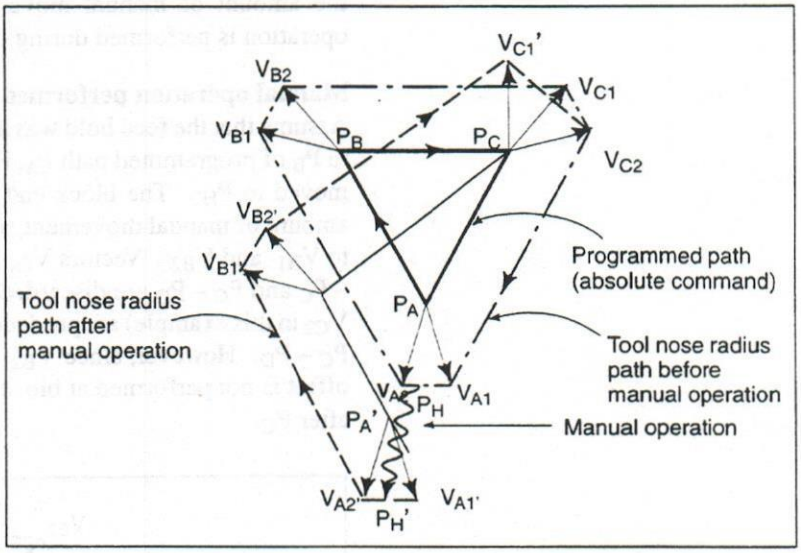
Manual operation performed in other than cornering

Assume that the feed hold was applied at point P_H while moving from P_A to P_B of programmed path P_A, P_B, and P_C and that the tool was manually moved to $P_{H'}$. The block end point P_B moves to the point $P_{B'}$ by the amount of manual movement, and vectors V_{B1} and V_{B2} at P_B also move to $V_{B1'}$ and $V_{B2'}$. Vectors V_{C1} and V_{C2} between the next two blocks $P_B - P_C$ and $P_C - P_D$ are discarded and new vectors $V_{C1'}$ and $V_{C2'}$ ($V_{C2'} = V_{C2}$ in this example) are produced from the relation between $P_{B'} - P_C$ and $P_C - P_D$. However, since $V_{B2'}$ is not a newly calculated vector, correct offset is not performed at block $P_{B'} - P_C$. Offset is correctly performed after P_C.



Manual operation during cornering

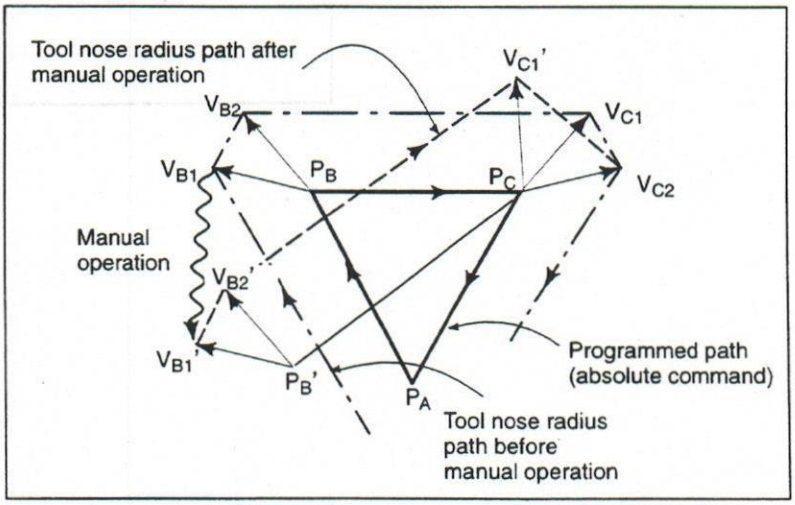
This is an example when manual operation is performed during cornering. $V_{A2'}$, $V_{B1'}$, and $V_{B2'}$ are vectors moved in parallel with V_{A2}, V_{B1} and V_{B2} by the amount of manual movement. The new vectors are calculated from V_{C1} and V_{C2}. Then correct tool nose radius compensation is performed for the blocks following Pc.



Manual operation after single block stop

Manual operation was performed when execution of a block was terminated by single block stop.

Vectors V_{B1} and V_{B2} are shifted by the amount of manual operation. Sub–sequent processing is the same as case a described above. An MDI operation can also be interveneted as well as manual operation. The movement is the same as that by manual operation.



4 AUTOMATIC OPERATION

Programmed operation of a CNC machine tool is referred to as automatic operation.
This chapter explains the following types of automatic operation:

- **MEMORY OPERATION**
  Operation by executing a program registered in CNC memory
- **MDI OPERATION (GUIDANCE PROGRAMMING MDI OPERATION)**
  Operation by executing a program entered from the MDI panel
- **DNC OPERATION**
  Function for operating a machine while a program from an extermal input/output unit.
- **MANUAL HANDLE INTERRUPTION**
  Function for performing manual feed during movement executed by automatic operation
- **MIRROR IMAGE**
  Function for enabling mirror–image movement along an axis during automatic operation
- **MANUAL INTERVENTION AND RETURN**
  Function restarting automatic operation by returning the tool the position where manual intervention was started during automatic operation.

4.1 MEMORY OPERATION

Programs are registered in memory in advance. When one of these programs is selected and the cycle start switch on the machine operator's panel is pressed, automatic operation starts, and the cycle start LED goes on.
When the feed hold switch on the machine operator's panel is pressed during automatic operation, automatic operation is stopped temporarily. When the cycle start switch is pressed again, automatic operation is restarted.
When the reset switch on the CRT/MDI panel is pressed, automatic operation terminates and the reset state is entered.
The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

Procedure for Memory Operation

1 Select a program to be executed according to either of the following procedure :

1–1 Selecting a program in **EDIT** mode

a Press the **EDIT** mode selection switch.

b Select the program to be executed from the registered programs, as follows :

(i) Press [PROG] to display the program list screen.

(ii) Position the cursor to the program to be executed by using the [↑] and [↓] cursor keys.

(iii) Instead of **(ii)**, you can also enter the program number directly by using the numeric keys.

(iv) Press the **[EDIT]** soft key.

c Press the MEMORY mode selection switch.

1–2 Selecting a program in **MEMORY** mode

a Press the **MEMORY** mode selection switch.

b Select the program to be executed from the registered programs, as follows:

(i) Press [PROG] to display the program screen.

(ii) Press the [O] address key.

(iii) Enter the program number by using the numeric keys.

(iv) Press the **[O SRH]** soft key.

2 Press the cycle start switch on the machine operator's panel. Automatic operation starts, and the cycle start LED goes on. When automatic operation terminates, the cycle start LED goes off.

3 To stop or cancel memory operation midway through, follow the steps below.

a. Stopping memory operation

Press the feed hold switch on the machine operator's panel. The feed hold LED goes on and the cycle start LED goes off. The machine responds as follows:

(i) When the machine was moving, feed operation decelerates and stops.

(ii) When dwell was being performed, dwell is stopped.

(iii) When M, S, or T was being executed, the operation is stopped after M, S, or T is finished.

When the cycle start switch on the machine operator's panel is pressed while the feed hold LED is on, machine operation restarts.

b. Terminating memory operation

Press the RESET key on the CRT/MDI panel.

Automatic operation is terminated and the reset state is entered. When a reset is applied during movement, movement decelerates then stops.

Explanation

Memory operation

After memory operation is started, the following are executed:

(1) A one–block command is read from the specified program.
(2) The block command is decoded.
(3) The command execution is started.
(4) The command in the next block is read.
(5) Buffering is executed. That is, the command is decoded to allow immediate execution.
(6) Immediately after the preceding block is executed, execution of the next block can be started. This is because buffering has been executed.
(7) Hereafter, memory operation can be executed by repeating the steps (4) to (6).

Stopping and terminating memory operation

Memory operation can be stopped using one of two methods: Specify a stop command, or press a key on the machine operator's panel.

- The stop commands include M00 (program stop), M01 (optional stop), and M02 and M30 (program end).
- There are two keys to stop memory operation: The feed hold key and reset key.

- **Program stop (M00)**

Memory operation is stopped after a block containing M00 is executed. When the program is stopped, all existing modal information remains unchanged as in single block operation. The memory operation can be restarted by pressing the cycle start button. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- **Optional stop (M01)**

Similarly to M00, memory operation is stopped after a block containing M01 is executed. This code is only effective when the Optional Stop switch on the machine operator's panel is set to ON. Operation may vary depending on the machine tool builder. Refer to the manual supplied by the machine tool builder.

- **Program end (M02, M30)**

When M02 or M30 (specified at the end of the main program) is read, memory operation is terminated and the reset state is entered.
In some machines, M30 returns control to the top of the program. For details, refer to the manual supplied by the machine tool builder.

- **Feed hold**

When Feed Hold button on the operator's panel is pressed during memory operation, the tool decelerates to a stop at a time.

- **Reset**

Automatic operation can be stopped and the system can be made to the reset state by using RESET key on the CRT/MDI panel or external reset signal. When reset operation is applied to the system during a tool moving status, the motion is slowed down then stops.

- **Optional block skip**

When the optional block skip switch on the machine operator's panel is turned on, blocks containing a slash (/) are ignored.

4.2 MDI OPERATION (GUIDANCE PROGRAMMING MDI OPERATION)

When the guidance programming function option is supported, you can create and execute a program by guidance programming in MDI mode. During guidance programming MDI operation, you can add a program that is executed by MDI operation to a program that is already registered in memory.
The following procedure is given as an example. For a full explanation of the actual operation, refer to the relevant manual supplied by the machine tool builder.

Procedure for MDI Operation

Procedure

1 Press the **MDI** mode selection switch.

2 Press the [PROG] key to display the program list screen. Pressing the right–most soft key on the program list screen changes the soft keys. Pressing the **[COMENT]** soft key hides the program names. To display the program names, press the **[COMENT]** soft key again (see Section 10.1.3 for details).

3 In response to the message "INPUT NO. OR SELECT MENU BY CURSOR" on the screen, enter the program number.

a When creating a new program

Enter a program number of up to four digits, by using the numeric keys.

b When adding an MDI operation program to a registered program

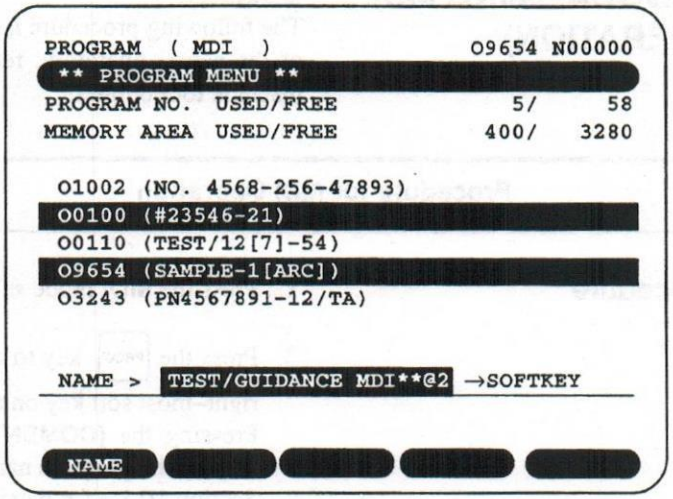
Enter the program number using the numeric keys, or select the program using the [↑] and [↓] cursor keys.

Example) When a new program number is entered

```
PROGRAM  ( MDI  )                    O9654 N00000
** PROGRAM MENU **
PROGRAM NO.  USED/FREE                 5/     58
MEMORY AREA  USED/FREE               480/   3280

 O1002 (NO. 4568-256-47893)
 O0100 (#23546-21)
 O0110 (TEST/12[7]-54)
 O9654 (SAMPLE-1[ARC])
 O3243 (PN4567891-12/TA)

 PROGRAM NO.   >  235
INPUT NO. OR SELECT MENU BY CURSOR          1/2
 EDIT
```

4 Press the **[EDIT]** soft key.

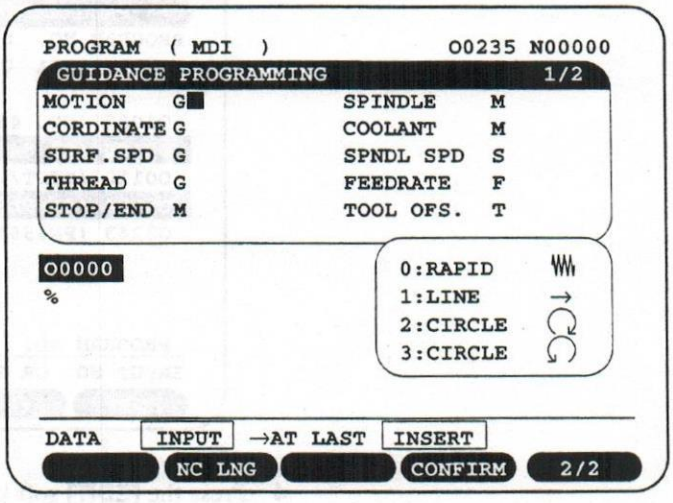
5 The **[NAME]** soft key and prompt to enter the program name appear. Enter a program name of up to 23 characters, using the alphanumeric keys, then press the **[NAME]** soft key. Note that the program name cannot include a semicolon (EOB).



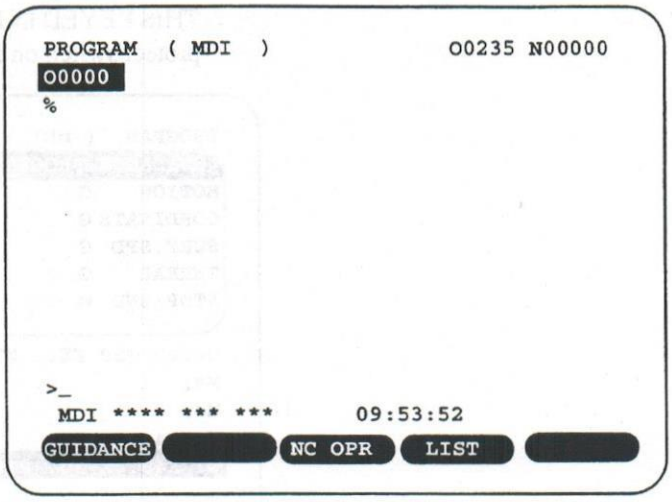
When a program name need not be registered, press the **[NAME]** soft key without entering any characters.
If you enter a previously registered program number in step 3, the prompt to enter a program name does not appear.

6 The guidance programming screen or CNC language screen appears, depending on the following parameter setting:
Bit 0 (NCE) of parameter No. 9320
0 : Displays the guidance programming screen once the program number has been registered.
1 : Displays the CNC language screen once the program number has been registered.

a When the NCE bit is set to 0, the guidance programming screen shown below appears. You can subsequently display the CNC language screen by pressing the **[NC LNG]** soft key.



b When the NCE bit is set to 1, the CNC language screen shown below appears. You can subsequently display the guidance programming screen by pressing the **[GUIDANCE]** soft key.



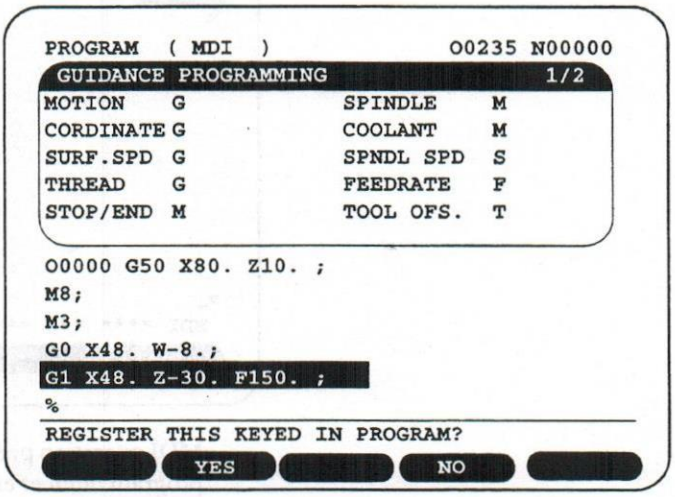
MDI operation program O0000 is displayed on the screen. The program number entered in step **3** is displayed at the top right of the screen. To return to the program list screen, press the **[LIST]** soft key.

7 Create a program by performing guidance programming. See Section 10.1.4 for details.
The guidance programming MDI operation allows up to nine blocks to be created at one time. When registering a program, created during guidance programming MDI operation, into memory, do not use M02 or M30 at the end of the program.

8 To execute the program displayed on the guidance programming screen, press the cycle start button on the machine operator's panel. The program is started.

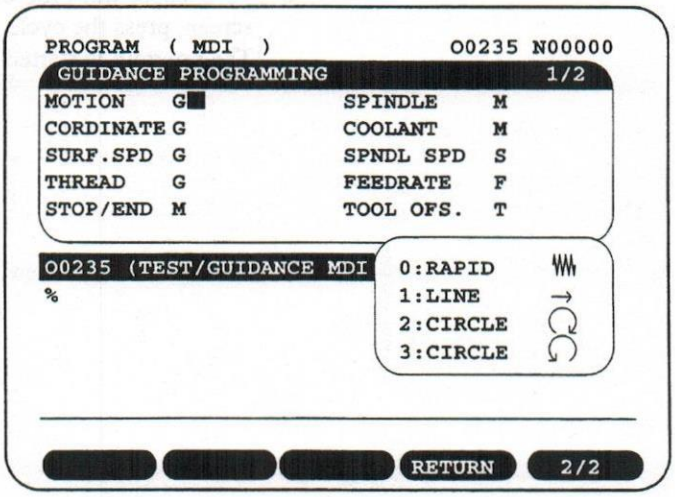
9 You can register the program executed by MDI operation into memory.

a Once the program has been executed, the message "REGISTER THIS KEYED IN PROGRAM?" appears, provided the memory protect switch on the machine operator's panel is released.

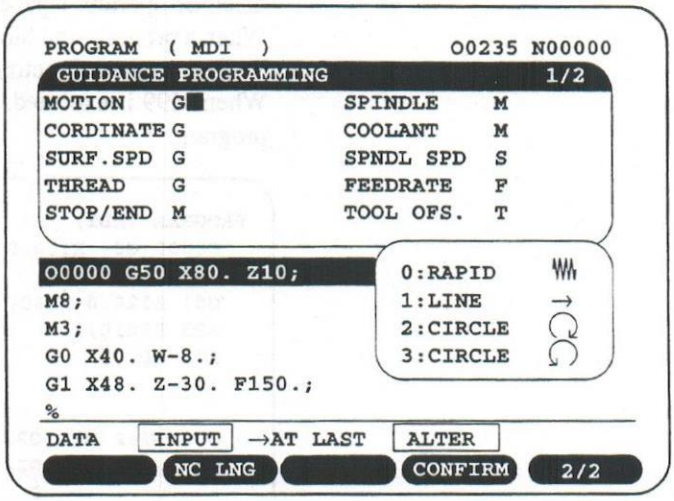


Pressing the **[YES]** soft key adds the program executed by the guidance programming MDI operation to the end of the program having the number entered in step 3. Pressing the **[NO]** soft key abandons registration of the program.

b You can display the contents of the program having the number entered in step 3 by pressing the **[CONFIRM]** soft key.
The cursor, used to indicate a block, can be moved with the page keys.



Pressing the **[RETURN]** soft key displays the contents of program O0000, the original program for guidance programming MDI operation.



c To continue MDI operation after registering the program into memory, repeat step **7** and subsequent steps.

10 To erase all programs created by MDI, press the RESET key. (In this case, bit 7 (MCL) of parameter No. 3203 must be set to 1.)

11 When stopping or terminating MDI operation, perform the following:

a Stopping MDI operation

Pressing the feed hold switch on the machine operator's panel turns on the feed hold LED and turns off the cycle start LED. The machine enters the following state:

(i) When the machine is moving the tool, the tool is decelerated until it stops.

(ii) During dwell, dwell is paused.

(iii) When the machine is performing an M, S, or T operation, the machine stops once the operation has been completed.

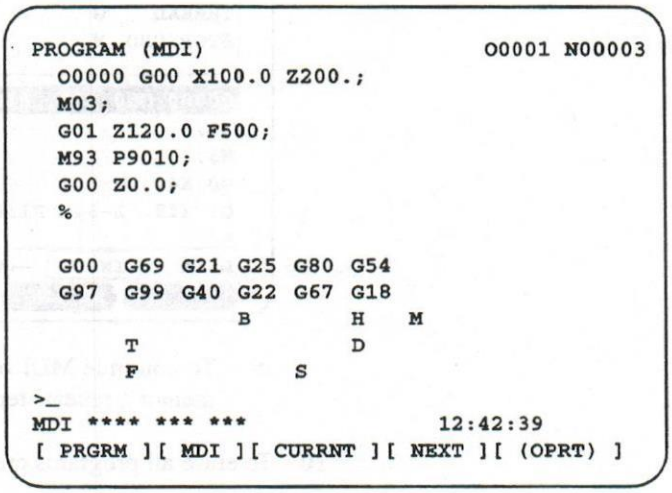
Pressing the cycle start switch on the machine operator's panel restarts the machine.

b Terminating MDI operation

Pressing the RESET key on the CRT/MDI panel terminates automatic operation and places the system in the reset state. When the system is reset while a tool is being moved, the tool is decelerated until it stops.

12 Pressing the **[NC OPR]** soft key enables programming with the CNC language. See Sections 9.2 to 9.7 for details of programming with the CNC language.

13 To execute the program displayed on the CNC language screen, position the cursor to the start of the program (or an arbitrary position within the program). Then, press the cycle start button on the machine operator's panel. The created program is started.
When a program end block (M02 or M03) or ER (%) is executed, the created program is automatically deleted, thus ending the operation. When M99 is executed, execution returns to the start of the created program.



Explanations

The previous explanation of how to execute and stop memory operation also applies to MDI operation, except that in MDI operation, M30 does not return control to the beginning of the program (M99 performs this function).

• Erasing the program

Programs prepared in the **MDI** mode will be erased in the following cases:

- In MDI operation, if M02, M30 or ER(%) is executed.
- In **MEMORY** mode, if memory operation is performed.
- In **EDIT** mode, if any editing is performed.
- Background editing is performed.
- Upon reset when bit 7 (MCL) of parameter No. 3203 is set to 1

• Restart

If the program is restarted during editing on the CNC language screen while the MDI operation is stopped, it is restarted from the line to which the cursor is positioned.
On the guidance programming screen, the program is always restarted from the first line.

Limitations

• Program registration

On the CNC language screen, you cannot register a program created by an MDI operation.

- **Number of lines in a program**

1. **On the guidance programming screen**
   A program of up to nine lines can be created.
2. **On the CNC language screen**
   A program consisting of up to six lines can be created. When parameter MDL (No. 3107 #7) is set to 0 to specify a mode that suppresses the display of continuous–state information, a program of up to 10 lines can be created.
   If the created program exceeds the specified number of lines, % (ER) is deleted (prevents insertion and modification).

- **Subprogram nesting**

Calls to subprograms (M98) can be specified in a program created in the **MDI** mode. This means that a program registered in memory can be called and executed during MDI operation. In addition to the main program executed by automatic operation, up to two levels of subprogram nesting are allowed (when the custom macro option is provided, up to four levels are allowed).

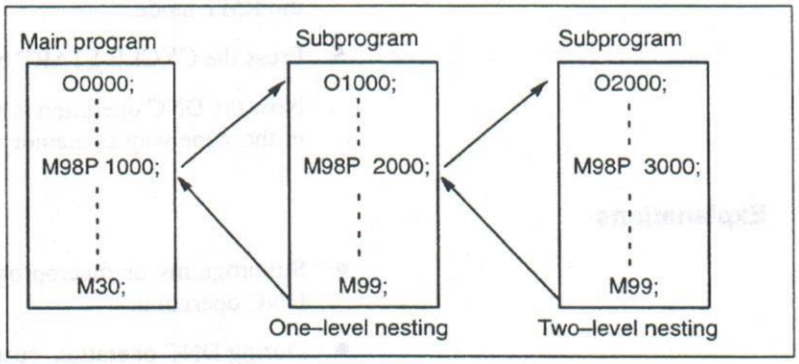


Fig. 4.2(a)Nesting Level of Subprograms Called from the MDI Program

- **Macro call**

When the custom macro option is provided, macro programs can also be created, called, and executed in the **MDI** mode. However, macro call commands cannot be executed when the mode is changed to **MDI** mode after memory operation is stopped during execution of a subprogram.

- **Memory area**

When a program is created in the **MDI** mode, an empty area in program memory is used. If program memory is full, no programs can be created in the **MDI** mode.

- **Editing a program during a guidance programming MDI operation**

Programs registered in memory (which can be displayed by pressing the **[CONFIRM]** soft key) cannot be edited during a guidance programming MDI operation. If you attempt to edit such a program, the message "CANNOT EDIT" is displayed.

Notes
If a program executed by a guidance programming MDI operation includes a miscellaneous function, such as M02 or M30, programs created in MDI mode may be cleared if the miscellaneous function resets the CNC. Whether programs created in MDI mode are cleared with a reset signal can be specifiedwith the following parameter. When the parameter is set to clear programs, programs cannot be registered into memory.
Bit 7 (MCL) of parameter No. 3203
0 : Programs created in MDI mode are not cleared upon reset.
1 : Programs created in MDI mode are cleared upon reset.

4.3 DNC OPERATION

DNC operation is a mode of machine operation in which the machine runs on a program being read directly from an external I/O device. The program need not reside in the CNC memory. This mode is useful when the program is too large to fit in the CNC memory.

Procedure for DNC operation

1 Select the MDI mode, and specify an I/O channel device channel connected to an I/O device on the setting screen.

2 Press the MEMORY switch (one of the mode selection switches).

3 Locate the beginning of the program on the I/O device.

4 Press the REMOTE switch on the machine operator's panel to select the RMT mode.

5 Press the CYCLE START button.

Now the DNC operation is activated. It can be stopped and restarted in the same way as memory operation.

Explanations

- Subprograms or macroprograms in memory can be called during DNC operation.
- During DNC operation, custom macros can be issued, but repetition or branch commands cannot be used.
- Macro programs in memory can be called during DNC operation.
- During DNC operation, a subprogram or macro program cannot make a return to the calling program by issuing a return command with a sequence number specified (M99P_).

4.4 MANUAL HANDLE INTERRUPTION

The movement by manual handle operation can be done by overlapping it with the movement by automatic operation in the automatic operation mode.

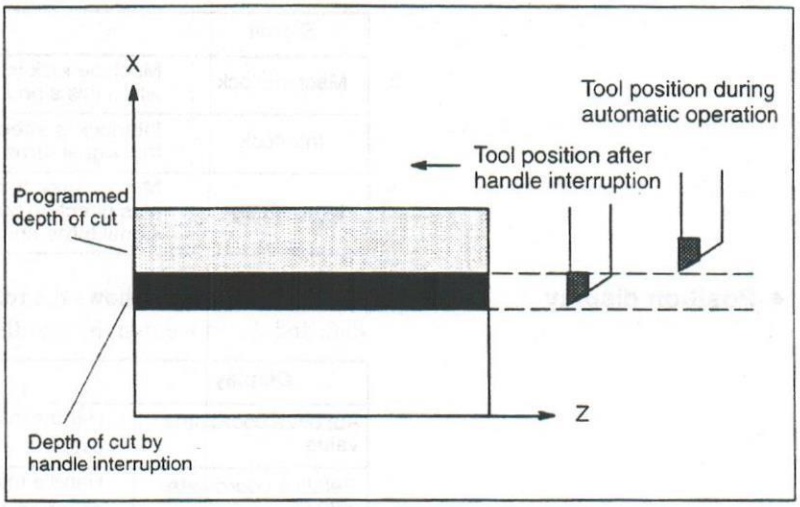


Fig. 4.4 (a) Manual Handle Interruption

·Handle interruption axis selection signals

For the handle interruption axis selection signals, refer to the manual supplied by the machine tool builder.

During automatic operation, handle interruption is enabled for an axis if the handle interruption axis selection signal for that axis is on. Handle interruption is performed by turning the handle of the manual pulse generator.

Notes

1. The travel distance by handle interruption is determined according to the amount by which the manual pulse generator is turned and the handle feed magnification (x1, x10, xM, xN). Since this movement is not accelerated or decelerated, it is very dangerous to use a large magnification value for handle interruption.
   The move amount per scale at x1 magnification is 0.001 mm (metric output) or 0.0001 inch (inch output).
2. Handle interruption is disabled when the machine is locked during automatic operation.

Explanations

• Relation with other functions

The following table indicates the relation between other functions and the movement by handle interrupt.

Signal	Relation
Machine lock	Machine lock is effective. The tool does not move even when this signal turns on.
Interlock	Interlock is effective. The tool does not move even when this signal turns on.
Mirror image	Mirror image is not effective. Interrupt functions on the plus direction by plus direction command, even if this signal turns on.

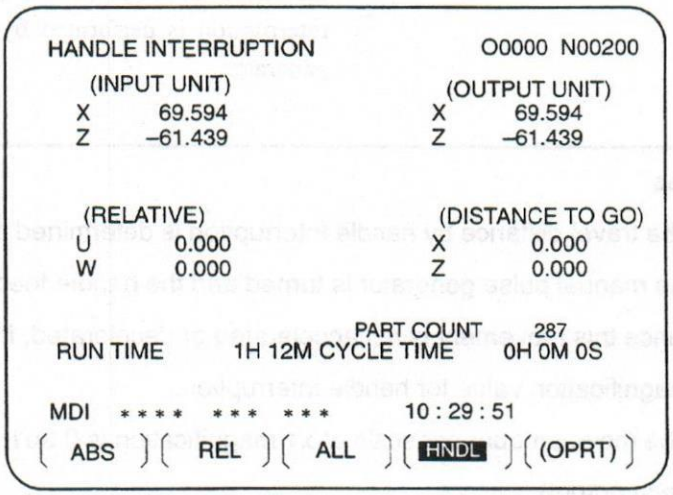
• Position display

The following table shows the relation between various position display data and the movement by handle interrupt.

Display	Relation
Absolute coordinate value	Handle interruption does not change absolute coordinates.
Relative coordinate value	Handle interruption does not change relative coordinates.
Machine coordinate value	Machine coordinates are changed by the travel distance specified by handle interruption.

• Travel distance display

Press the [POS] function key, then press the HNDL chapter selection soft key. The distance moved by the tool as a result of the handle interrupt is displayed. The following 4 kinds of data are displayed concurrently.



(a) INPUT UNIT : Handle interrupt move amount in input unit system
Indicates the travel distance specified by handle interruption according to the least input increment.

(b) OUTPUT UNIT : Handle interrupt move amount in output unit system
Indicates the travel distance specified by handle interruption according to the least command increment.

(c) RELATIVE : Position in relative coordinate system
These values have no effect on the travel distance specified by handle interruption.

(d) DISTANCE TO GO : The remaining travel distance in the current block has no effect on the travel distance specified by handle interruption.

Display of the handle interrupt move amount is cleared when the manual reference position return ends every axis. Also, it is cleared by the soft key **[CANCEL]**.

4.5 MIRROR IMAGE

During automatic operation, the mirror image function can be used for movement along an axis. To use this function, set the mirror image switch to ON on the machine operator's panel, or set the mirror image setting to ON from the CRT/MDI panel.

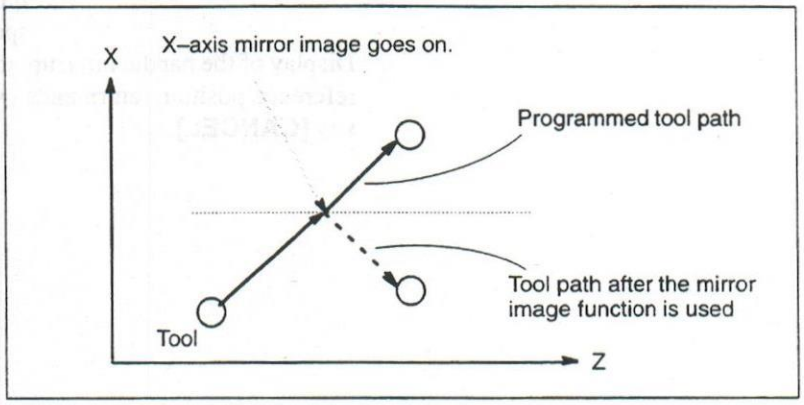


Fig. 4.5 (a) Mirror Image

Procedure

The following procedure is given as an example. For actual operation, refer to the manual supplied by the machine tool builder.

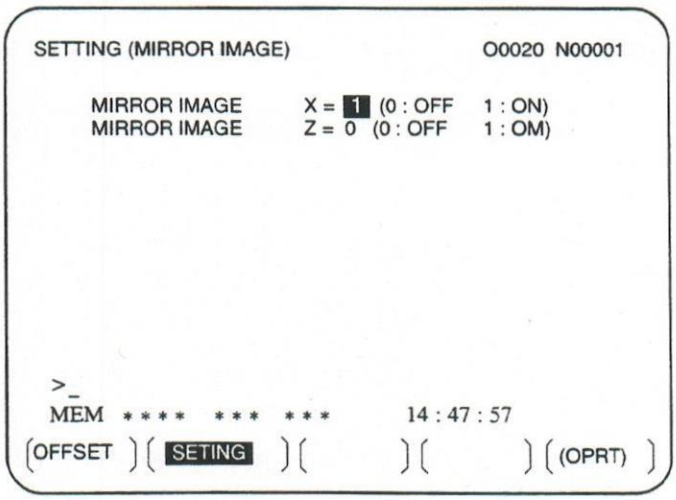
1 Press the single block switch to stop automatic operation. When the mirror image function is used from the beginning of operation, this step is omitted.

2 Press the mirror image switch for the target axis on the machine operator's panel.
Alternatively, turn on the mirror image setting by following the steps below:

2–1 Set the **MDI** mode.

2–2 Press the [OFFSET SETTING] function key.

2–3 Press the **[SETING]** soft key for chapter selection to display the setting screen.



2–4 Move the cursor to the mirror image setting position, then set the target axis to 1.

3 Enter an automatic operation mode (memory mode or MDI mode), then press the cycle start button to start automatic operation.

Explanations

- The mirror image function can also be turned on and off by setting bit 0 of parameter 0012 (MIRx) to 1 (on) or 0 (off).
- For the mirror image switches, refer to the manual supplied by the machine tool builder.

Restrictions

The direction of movement during manual operation, the direction of movement from an intermediate point to the reference position during automatic reference position return (G28).

4.6 MANUAL INTERVENTION AND RETURN

In cases such as when tool movement along an axis is stopped by feed hold during automatic operation so that manual intervention can be used to replace the tool: When automatic operation is restarted, this function returns the tool to the position where manual intervention was started. To use the conventional program restart function and tool withdrawal and return function, the switches on the operator's panel must be used in conjunction with the MDI keys. This function does not require such operations.

Explanations

- **Manual absolute on/off**

  In manual absolute off mode, the tool does not return to the stop position, but instead operates according to the manual absolute on/off function.

- **Override**

  For the return operation, the dry run feedrate is used, and the jog feedrate override function is enabled.

- **Return operation**

  Return operation is performed according to positioning based on nonlinear interpolation.

- **Single block**

  If the single block stop switch is on during return operation, the tool stops at the stop position and restarts movement when the cycle start switch is pressed.

- **Cancellation**

  If a reset occurs or an alarm is issued during manual intervention or the return operation, this function is cancelled.

- **MDI mode**

  This function can be used in the MDI mode as well.

Restrictions

- **Enabling and disabling manual intervention and return**

  This function is enabled only when the automatic operation hold LED is on. When there is no travel distance remaining, this function has no effect even if a feed hold stop is performed with the automatic operation hold signal *SP (bit 5 of G008).

- **Offset**

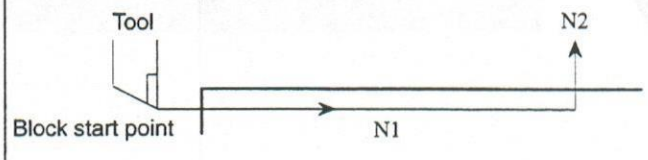
  When the tool is replaced using manual intervention for a reason such as damage, the tool movement cannot be restarted by a changed offset in the middle of the interrupted block.

- **Machine lock and mirror image**

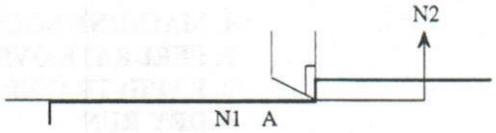
  When performing manual intervention, never use the machine lock or mirror image.

Example

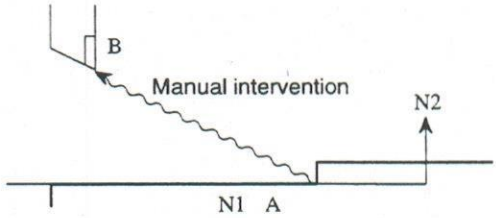
1. The N1 block cuts a workpiece



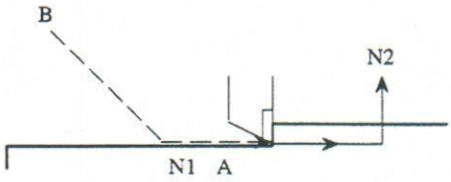
2. The tool is stopped by pressing the feed hold switch in the middle of the N1 block (point A).



3. After retracting the tool manually to point B, tool movement is restarted.



4. After automatic return to point A at the dry run feedrate, the remaining move command of the N1 block is executed.



Notes

When performing manual intervention, pay particular attention of machining and the shape of the workpiece so that the machine and tool are not damaged.

5 TEST OPERATION

The following functions are used to check before actual machining whether the machine operates as specified by the created program.

1. MACHINE LOCK AND AUXILIARY FUNCTION LOCK
2. FEEDRATE OVERRIDE
3. RAPID TRAVERSE OVERRIDE
4. DRY RUN
5. SINGLE BLOCK

5.1 MACHINE LOCK AND AUXILIARY FUNCTION LOCK

To display the change in the position without moving the tool, use machine lock.
There are two types of machine lock, all–axis machine lock, which stops the movement along all axes, and specified–axis machine lock, which stops the movement along specified axes only. In addition, auxiliary function lock, which disables M, S, and T commands, is available for checking a program together with machine lock.

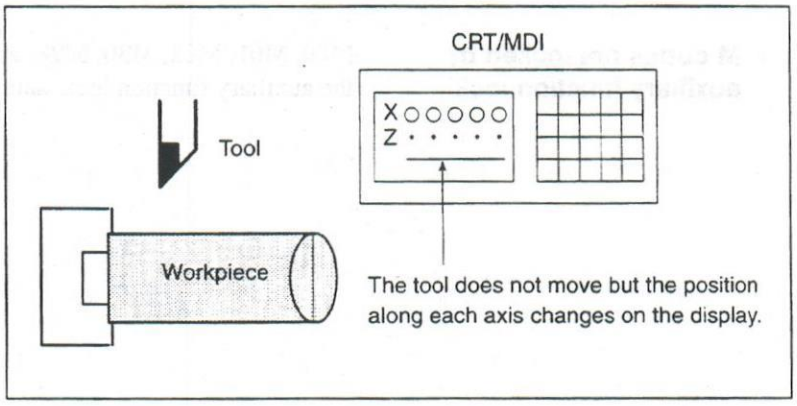


Fig. 5.1 Machine lock

Procedure for Machine Lock and Auxiliary Function Lock

- **Machine Lock**

Press the machine lock switch on the operator's panel. The tool does not move but the position along each axis changes on the display as if the tool were moving.
Some machines have a machine lock switch for each axis. On such machines, press the machine lock switches for the axes along which the tool is to be stopped. Refer to the appropriate manual provided by the machine tool builder for machine lock.
The position relationship between the workpiece coordinates and machine coordinates may change after automatic operation by the machine lock function has been executed. If this occurs, reset the workpiece coordinate system by specifying the coordinate system setting command or by making a manuale reference position return.

- **Auxiliary Function Lock**

Press the auxiliary function lock switch on the operator's panel. M, S, and T codes are disabled and not executed. Refer to the appropriate manual provided by the machine tool builder for auxiliary function lock.

Restrictions

- **M, S, T command by only machine lock**

  M, S, and T commands are executed in the machine lock state.

- **Reference position return under Machine Lock**

  When a G27, G28, or G30 command is issued in the machine lock state, the command is accepted but the tool does not move to the reference position and the reference position return LED does not go on.

- **M codes not locked by auxiliary function lock**

  M00, M01, M02, M30, M98, and M99 commands are executed even in the auxiliary function lock state.

5.2 FEEDRATE OVERRIDE

A programmed feedrate can be reduced or increased by a percentage (%) selected by the override dial. This feature is used to check a program. For example, when a feedrate of 100 mm/min is specified in the program, setting the override dial to 50% moves the tool at 50 mm/min.

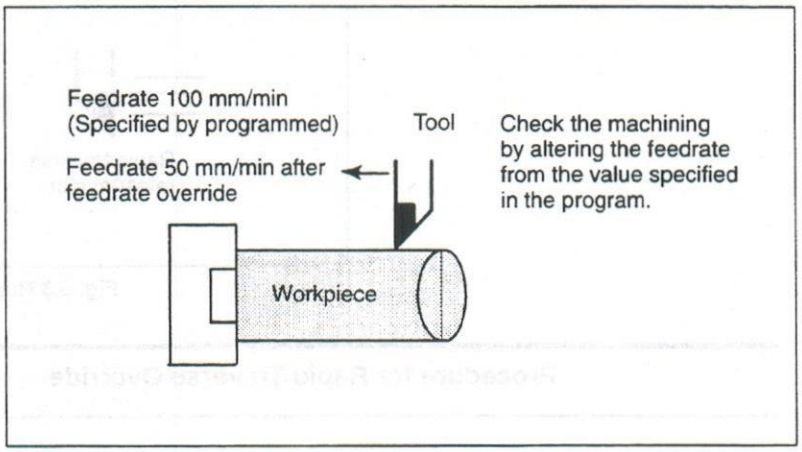
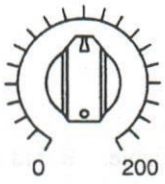

Feedrate 100 mm/min
(Specified by programmed)
Tool
Check the machining by altering the feedrate from the value specified in the program.
Feedrate 50 mm/min after feedrate override
Workpiece

Fig. 5.2 Feedrate override

Procedure for Feedrate Override



FEED RATE OVERRIDE

Set the feedrate override dial to the desired percentage (%) on the machine operator's panel, before or during automatic operation.
On some machines, the same dial is used for the feedrate override dial and jog feedrate dial. Refer to the appropriate manual provided by the machine tool builder for feedrate override.

Restrictions

- **Override Range**

  The override that can be specified ranges from 0 to 254%. For individual machines, the range depends on the specifications of the machine tool builder.

- **Override during thread**

  During threading, the override is ignored and the feedrate remains as specified by program.

5.3 RAPID TRAVERSE OVERRIDE

An override of four steps (F0, 25%, 50%, and 100%) can be applied to the rapid traverse rate. F0 is set by a parameter (No. 1421).

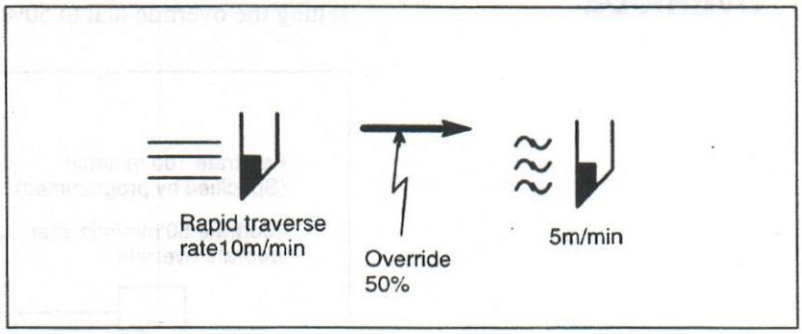
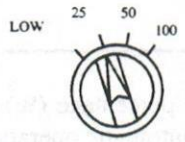


Fig. 5.3 Rapid traverse override

Procedure for Rapid Traverse Override

Select one of the four feedrates with the rapid traverse override switch during rapid traverse. Refer to the appropriate manual provided by the machine tool builder for rapid traverse override.



Rapid traverse override

Explanation

The following types of rapid traverse are available. Rapid traverse override can be applied for each of them.

1) Rapid traverse by G00.
2) Rapid traverse during a canned cycle.
3) Rapid traverse in G27, G28 and G30.
4) Manual rapid traverse.
5) Rapid traverse of manual reference position return

5.4 DRY RUN

The tool is moved at the feedrate specified by a parameter regardless of the feedrate specified in the program. This function is used for checking the movement of the tool under the state that the workpiece is removed from the table.

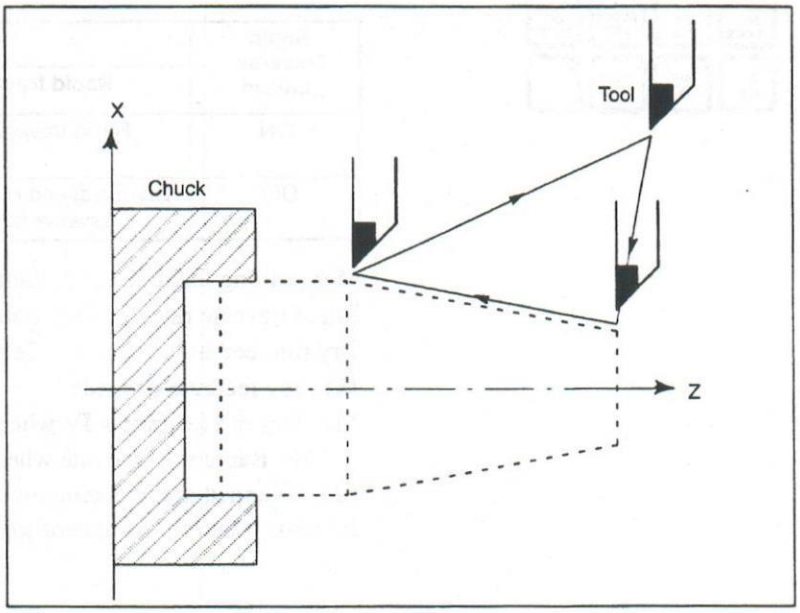


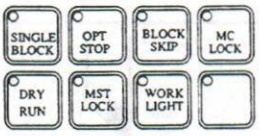
Fig. 5.4 Dry run

Procedure for Dry Run

Press the dry run switch on the machine operator's panel during automatic operation.
The tool moves at the feedrate specified in a parameter. The rapid traverse switch can also be used for changing the feedrate.
Refer to the appropriate manual provided by the machine tool builder for dry run.

Explanation

- **Dry run feedrate**



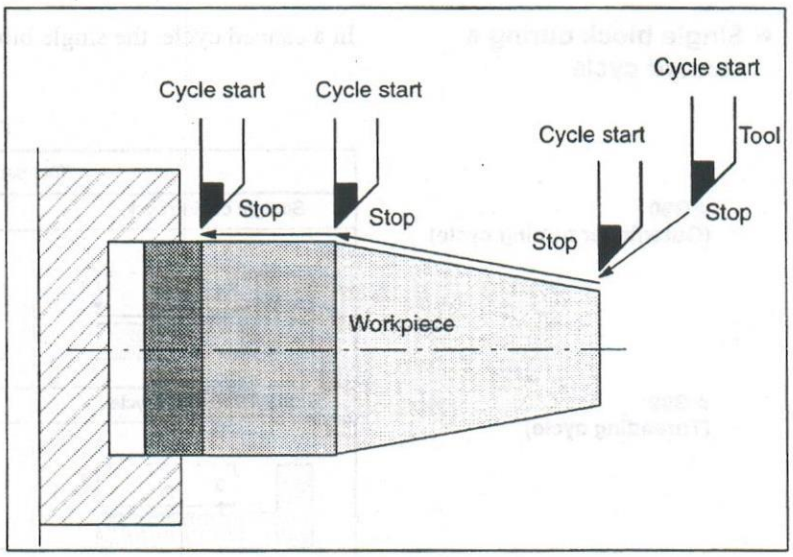
The dry run feedrate changes as shown in the table below according to the rapid traverse switch and parameters.

Rapid traverse button	Program command	
	Rapid traverse	Feed
ON	Rapid traverse rate	Dry run feedrate × Max. JV *2)
OFF	Dry run speed × JV,or rapid traverse rate *1)	Dry run feedrate × JV *2)

Max. cutting feedrate . . . . . Setting by parameter No.1422
Rapid traverse rate . . . . . Setting by parameter No.1420
Dry run feedrate . . . . . Setting by parameter No.1410
JV: Jog feedrate override
*1: Dry run feedrate x JV when parameter RDR (bit 6 of No. 1401) is 1. Rapid traverse rate when parameter RDR is 0.
*2: Clamped at the maximum cutting feedrate.
JV max: Maximum value of jog feedrate override.

5.5 SINGLE BLOCK

Pressing the single block switch starts the single block mode. When the cycle start button is pressed in the single block mode, the tool stops after a single block in the program is executed. Check the program in the single block mode by executing the program block by block.



Single block

Procedure for Single Block

1 Press the single block switch on the machine operator's panel. The execution of the program is stopped after the current block is executed.

2 Press the cycle start button to execute the next block. The tool stops after the block is executed.

Refer to the appropriate manual provided by the machine tool builder for single block execution.

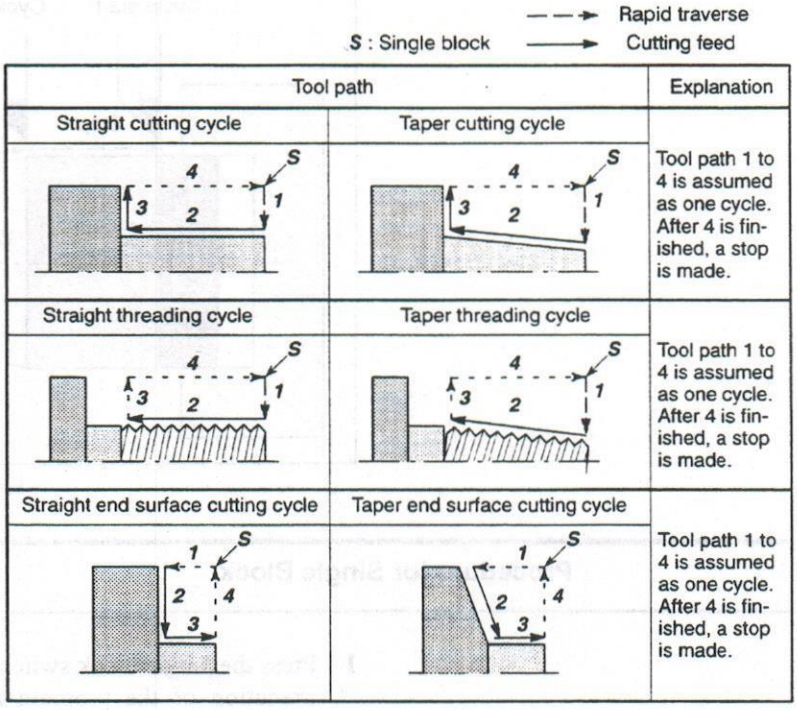
Explanation

- **Reference position return and single block**

If G28 to G30 are issued, the single block function is effective at the intermediate point.

- **Single block during a canned cycle**

In a canned cycle, the single block stop points are as follows.


Rapid traverse
S : Single block
Cutting feed
Tool path
Explanation
Straight cutting cycle
Taper cutting cycle
Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
Straight threading cycle
Taper threading cycle
Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.
Straight end surface cutting cycle
Taper end surface cutting cycle
Tool path 1 to 4 is assumed as one cycle. After 4 is finished, a stop is made.

☆G90 (Outer/inner turning cycle)

☆G92 (Threading cycle)

☆G94 (End surface turning cycle)

Fig. 5.5(a) Single block during canned cycle

- **Macro call and single block**

Single block stop is not performed in the G65 block.

6 SAFETY FUNCTIONS

To immediately stop the machine for safety, press the Emergency stop button. To prevent the tool from exceeding the stroke ends, Overtravel check and Stroke check are available. This chapter describes emergency stop, overtravel check, and stroke check.

6.1 EMERGENCY STOP

If you press Emergency Stop button on the machine operator's panel, the machine movement stops in a moment.

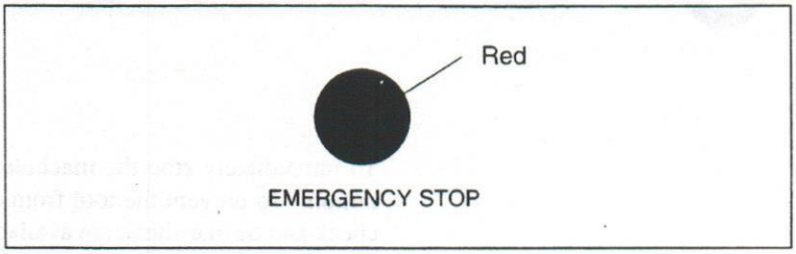


Fig. 6.1 Emergency stop

This button is locked when it is pressed. Although it varies with the machine tool builder, the button can usually be unlocked by twisting it.

Explanation

EMERGENCY STOP interrupts the current to the motor.
Causes of trouble must be removed before the button is released.

6.2 OVERTRAVEL

When the tool tries to move beyond the stroke end set by the machine tool limit switch, the tool decelerates and stops because of working the limit switch and an OVER TRAVEL is displayed.

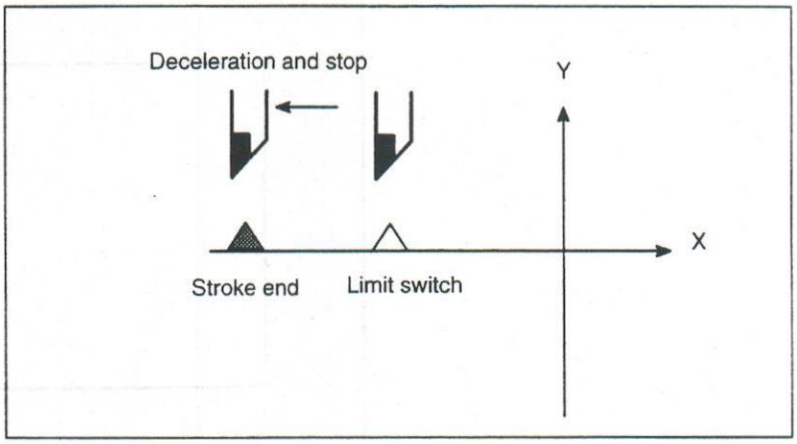


Fig. 6.2 Overtravel

Explanation

- **Overtravel during automatic operation**

When the tool touches a limit switch along an axis during automatic operation, the tool is decelerated and stopped along all axes and an overtravel alarm is displayed.

- **Overtravel during manual operation**

In manual operation, the tool is decelerated and stopped only along the axis for which the tool has touched a limit switch. The tool still moves along the other axes.

- **Releasing overtravel**

Press the reset button to reset the alarm after moving the tool to the safety direction by manual operation. For details on operation, refer to the operator's manual of the machine tool builder.

- **Alarm**

No.	Message	Description
506	Overtravel: +n	The tool has exceeded the hardware–specified overtravel limit along the positive nth axis (n: 1 to 8).
507	Overtravel: –n	The tool has exceeded the hardware–specified overtravel limit along the negative nth axis (n: 1 to 8).

6.3 STROKE CHECK

There areas which the tool cannot enter can be specified with stored stroke limit 1.

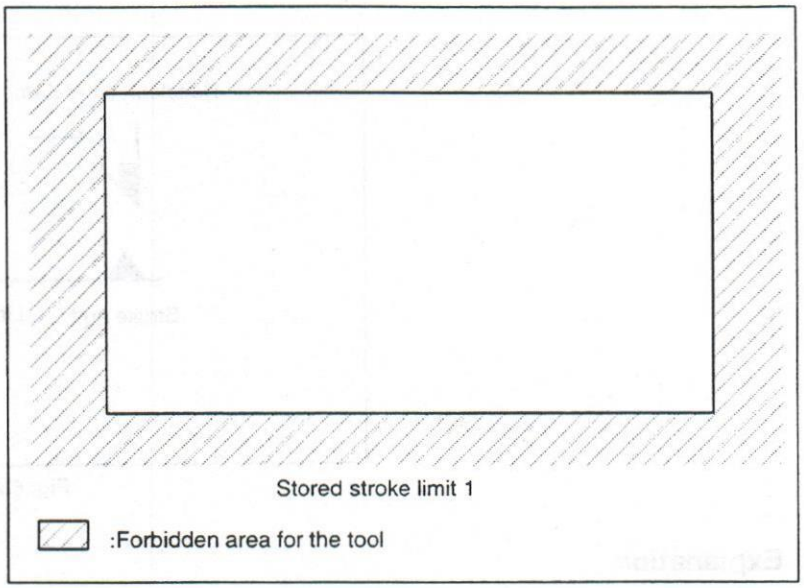


Fig. 6.3 (a) Stroke check

When the tool exceeds a stored stroke limit, an alarm is displayed and the tool is decelerated and stopped.
When the tool enters a forbidden area and an alarm is generated, the tool can be moved in the reverse direction from which the tool came.

Explanation

- **Stored stroke limit 1**

Parameters (Nos. 1320, 1321 or Nos. 1326, 1327) set boundary. Outside the area of the set limits is a forbidden area. The machine tool builder usually sets this area as the maximum stroke.

- **Checkpoint for the forbidden area**

The parameter setting or programmed value (XZIK) depends on which part of the tool or tool holder is checked for entering the forbidden area. Confirm the checking position (the top of the tool or the tool chuck) before programming the forbidden area.
If point C (The top of the tool) is checked in Fig. 6.3 (b) , the distance "c" should be set as the data for the stored stroke limit function. If point D (The tool chuck) is checked, the distance "d" must be set.

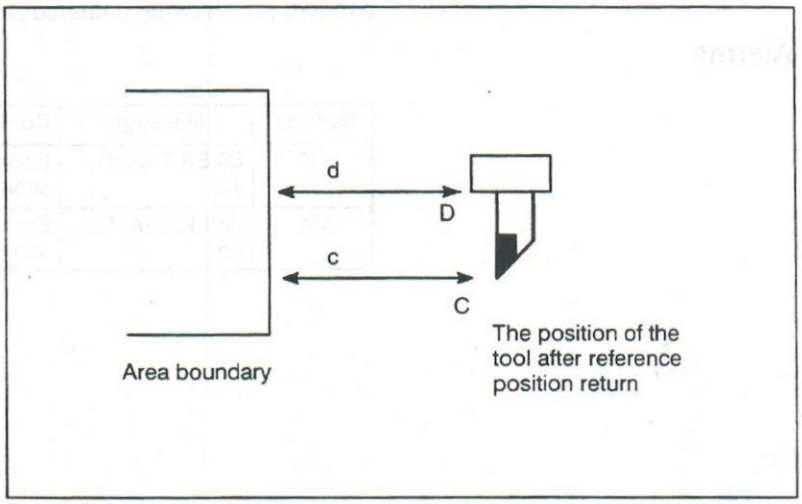


Fig. 6.3 (b) Setting the forbidden area

- **Effective time for a forbidden area**

Each limit becomes effective after the power is turned on and manual reference position return or automatic reference position return by G28 has been performed.
After the power is turned on, if the reference position is in the forbidden area of the limit, an alarm is generated immediately.

- **Releasing the alarms**

If the tool enters a forbidden area and an alarm is generated, the tool can be moved only in the backward direction. To cancel the alarm, move the tool backward until it is outside the forbidden area and reset the system. When the alarm is canceled, the tool can be moved both backward and forward.

- **Setting the forbidden area for the 16–TB (two–path control)**

For the 16–TB (two–path control), set a forbidden area for each tool post.

Notes
In setting a forbidden area, if the two points to be set are the same, the area is as follows:
(1)When the forbidden area is limit 1, all areas are forbidden areas.

- **Overrun amount of stored stroke limit**

If the rapid traverse rate is F (mm/minute), the maximum overrun amount, L mm, of the stored stroke limit is obtained from the following expression:

L (mm) = F/7500

The tool enters the specified inhibited area by up to L mm. The tool can be stopped at a position up to L mm in front of the inhibited area if parameter BFA (No.1300#7) is specified accordingly. In this case, the tool will not enter the inhibited area.

Alarms

Number	Message	Contents
500	OVER TRAVEL: +n	Exceeded the n–th axis (1–2) + side stored stroke limit 1.
501	OVER TRAVEL: –n	Exceeded the n–th axis (1–2) – side stored stroke limit 1.

7 ALARM AND SELF–DIAGNOSIS FUNCTIONS

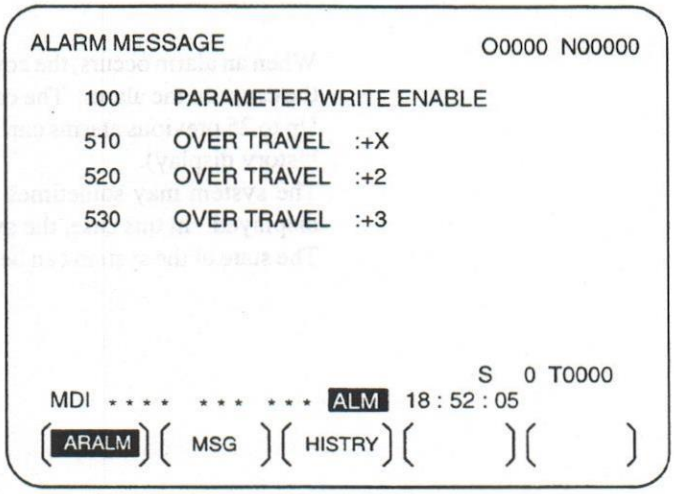
When an alarm occurs, the corresponding alarm screen appears to indicate the cause of the alarm. The causes of alarms are classified by error codes. Up to 25 previous alarms can be stored and displayed on the screen (alarm history display).
The system may sometimes seem to be at a halt, although no alarm is displayed. In this case, the system may be performing some processing. The state of the system can be checked using the self–diagnostic function.

7.1 ALARM DISPLAY

Explanations

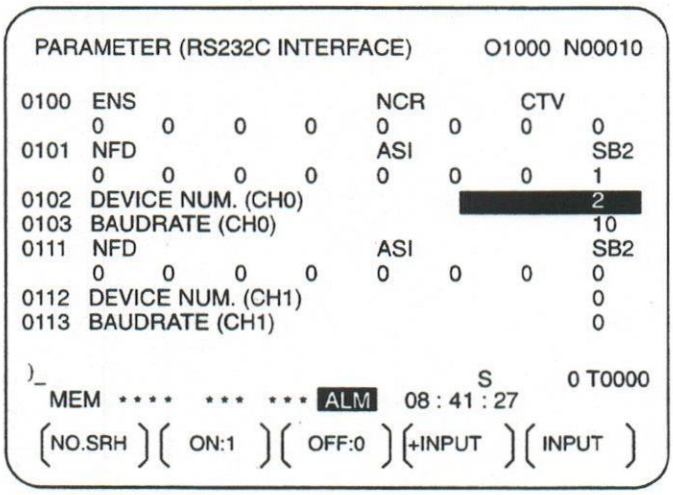
- **Alarm screen**

When an alarm occurs, the alarm screen appears.



- **Another method for alarm displays**

In some cases, the alarm screen does not appear, but an ALM is displayed at the bottom of the screen.


PARAMETER (RS232C INTERFACE) O1000 N00010
0100 ENS NCR CTV
0 0 0 0 0 0 0 0
0101 NFD ASI SB2
0 0 0 0 0 0 0 1
0102 DEVICE NUM. (CH0) 2
0103 BAUDRATE (CH0) 10
0111 NFD ASI SB2
0 0 0 0 0 0 0 0
0112 DEVICE NUM. (CH1) 0
0113 BAUDRATE (CH1) 0
S 0 T0000
MEM **** *** *** ALM 08 : 41 : 27
NO.SRH
ON:1
OFF:0
+INPUT
INPUT

In this case, display the alarm screen as follows:

1. Press the function key MESSAGE.
2. Press the chapter selection soft key **[ALARM]**.

- **Reset of the alarm**

Error codes and messages indicate the cause of an alarm. To recover from an alarm, eliminate the cause and press the reset key.

- **Error codes**

The error codes are classified as follows:
No. 000 to 255, 5000 to: Program errors(*)
No. 300 to 349: Absolute pulse coder (APC) alarms
No. 350 to 399: Serial pulse coder (SPC) alarms
No. 400 to 499: Servo alarms
No. 500 to 599: Overtravel alarms
No. 700 to 704: Overheat alarms
No. 749 to 799: Spindle alarms
No. 900 to 999: System alarms

*For an alarm (No. 000 to 255) that occurs in association with background operation, the indication "xxxBP/S alarm" is provided (where xxx is an alarm number). Only a BP/S alarm is provided for No. 140.
See the error code list in the APPENDIX G for details of the error codes.

7.2 ALARM HISTORY DISPLAY

Up to 25 of the most recent CNC alarms are stored and displayed on the screen.
Display the alarm history as follows:

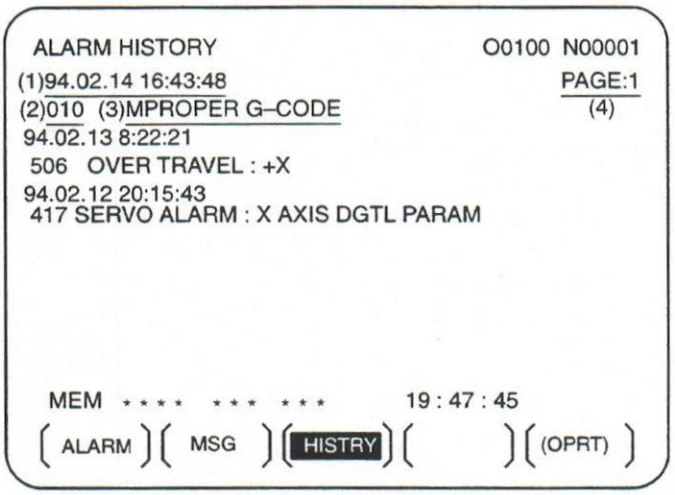
Procedure for Alarm History Display

1 Press the function key [MESSAGE].

2 Press the chapter selection soft key **[HISTRY]**.
The alarm history appears.
The following information items are displayed.
(1)The date the alarm was issued
(2)Alarm No.
(3)Alarm message (some contains no message)

3 You can change the page by pressing a page key.

4 To delete the recorded information, press the softkey **[(OPRT)]** then the **[DELETE]** key.



(1)The date the alarm was issued
(2)Alarm No.
(3)Alarm message (some contains no message)
(4)Page No.

7.3 CHECKING BY SELF–DIAGNOSTIC SCREEN

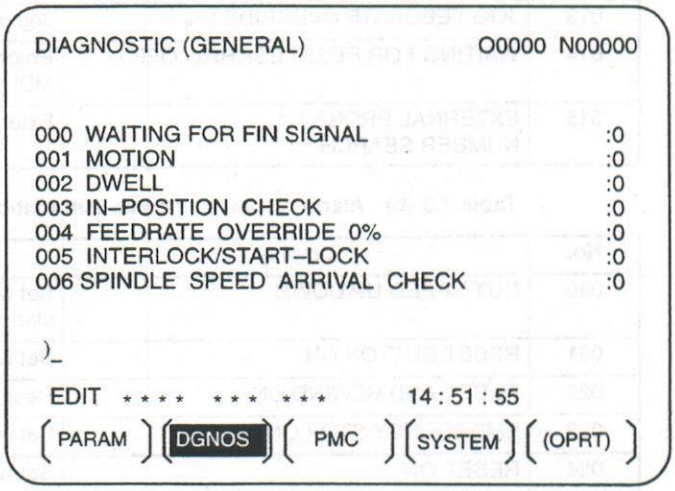
The system may sometimes seem to be at a halt, although no alarm has occurred. In this case, the system may be performing some processing. The state of the system can be checked by displaying the self–diagnostic screen.

Procedure for Diagnostic

1 Press the function key [SYSTEM].

2 Press the chapter select key **[DGNOS]**.

3 The diagnostic screen has more than 1 pages. Select the screen by the following operation.
   (1) Change the page by the 1–page change key.
   (2) Method by soft key
   – Key input the number of the diagnostic data to be displayed.
   – Press **[N SRCH]**.


DIAGNOSTIC (GENERAL) O0000 N00000
000 WAITING FOR FIN SIGNAL :0
001 MOTION :0
002 DWELL :0
003 IN–POSITION CHECK :0
004 FEEDRATE OVERRIDE 0% :0
005 INTERLOCK/START–LOCK :0
006 SPINDLE SPEED ARRIVAL CHECK :0
EDIT **** *** *** 14 : 51 : 55
PARAM DGNOS PMC SYSTEM (OPRT)

Explanations

Diagnostic numbers 000 to 015 indicate states when a command is being specified but appears as if it were not being executed. The table below lists the internal states when 1 is displayed at the right end of each line on the screen.

Table 7.3 (a) Alarm displays when a command is specified but appears as if it were not being executed

No.	Display	Internal status when 1 is displayed
000	WAITING FOR FIN SIGNAL	M, S. T function being executed
001	MOTION	Move command in automatic operation being executed
002	DWELL	Dwell being executed
003	IN–POSITION CHECK	In–position check being executed
004	FEEDRATE OVERRIDE 0%	Cutting feed override 0%
005	INTERLOCK/START–LOCK	Interlock ON
006	SPINDLE SPEED ARRIVAL CHECK	Waiting for spindle speed arrival signal to turn on
010	PUNCHING	Data being output via reader puncher interface
011	READING	Data being input via reader puncher interface
012	WAITING FOR (UN) CLAMP	0 is always displayed. (unused)
013	JOG FEEDRATE OVERRIDE 0%	Jog override 0%
014	WAITING FOR RESET.ESP.RRW.OFF	Emergency stop, external reset, reset & rewind, or MDI panel reset key on
015	EXTERNAL PROGRAM NUMBER SEARCH	External program number searching

Table 7.3 (b) Alarm displays when an automatic operation is stopped or paused.

No.	Display	Internal status when 1 is displayed
020	CUT SPEED UP/DOWN	Set when emergency stop turns on or when servo alarm occurs
021	RESET BUTTON ON	Set when reset key turns on
022	RESET AND REWIND ON	Reset and rewind turned on
023	EMERGENCY STOP ON	Set when emergency stop turns on
024	RESET ON	Set when external reset, emergency stop, reset, or reset & rewind key turns on
025	STOP MOTION OR DWELL	A flag which stops pulse distribution. It is set in the following cases. (1)External reset turned on. (2)Reset & rewind turned on. (3)Emergency stop turned on. (4)Feed hold turned on. (5)The MDI panel reset key turned on. (6)Switched to the manual mode (JOG/HANDLE/INC). (7)Other alarm occurred. (There is also alarm which is not set.)

Diagnostic numbers 020 to 025 indicate the states when automatic operation is stopped or paused.

The table below shows the signals and states which are enabled when each diagnostic data item is 1. Each combination of the values of the diagnostic data indicates a unique state.

020	CUT SPEED UP/DOWN	1	0	0	0	1	0	0
021	RESET BUTTON ON	0	0	1	0	0	0	0
022	RESET AND REWIND ON	0	0	0	1	0	0	0
023	EMERGENCY STOP ON	1	0	0	0	0	0	0
024	RESET ON	1	1	1	1	0	0	0
025	STOP MOTION OR DWELL	1	1	1	1	1	1	0

Emergency stop signal input
External reset signal input
MDI reset button turned on
Reset & rewind input
Servo alarm generation
Changed to another mode or feed hold
Single block stop

Diagnostic numbers 030 and 031 indicate TH alarm states.

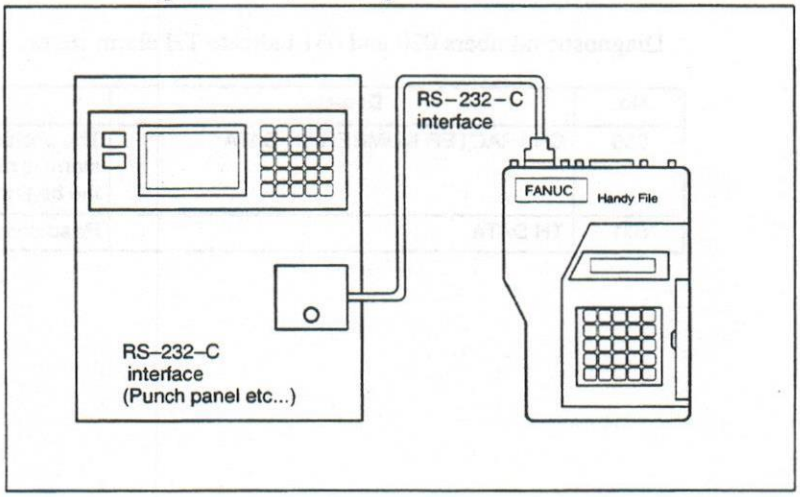
No.	Display	Meaning of data
030	CHARACTER NUMBER TH DATA	The position of the character which caused TH alarm is displayed by the number of characters from the beginning of the block at TH alarm
031	TH DATA	Read code of character which caused TH alarm

8 DATA INPUT/OUTPUT

NC data is transferred between the CNC and external input/output devices such as the Handy File.
The following types of data can be entered and output :

1.Program
2.Offset data
3.Parameter
4.Pitch error compensation data
5.Custom macro common variable

Before an input/output device can be used, the input/output related parameters must be set.
For how to set parameters, see Chapter 2**VOPERATIONAL DEVICESW**.



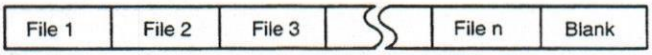
8.1 FILES

Of the external input/output devices, the FANUC Handy File and FANUC Floppy Cassette use floppy disks as their input/output medium, and the FANUC FA Card uses an FA card as its input/output medium.
In this manual, these input/output medium is generally referred to as a floppy. However, when the description of one input/output medium varies from the description of another, the name of the input/output medium is used. In the text below, a floppy represents a floppy disk or FA card.
Unlike an NC tape, a floppy allows the user to freely choose from several types of data stored on one medium on a file–by–file basis.
Input/output is possible with data extending over more than one floppy disk.

Explanations

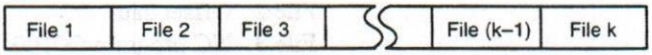
• What is a File

The unit of data, which is input/output between the floppy and the CNC by one input/output operation (pressing the VREADW or VPUNCHW key), is called a Hfilel. When inputting CNC programs from, or outputting them to the floppy, for example, one or all programs within the CNC memory are handled as one file.
Files are assigned automatically file numbers 1,2,3,4 and so on, with the lead file as 1.

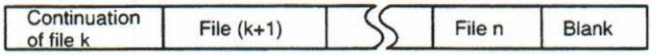


• Request for floppy replacement

When one file has been entered over two floppies, LEDs on the adaptor flash alternately on completion of data input/output between the first floppy and the CNC, prompting floppy replacement. In this case, take the first floppy out of the adaptor and insert a second floppy in its place. Then, data input/output will continue automatically.
Floppy replacement is prompted when the second floppy and later is required during file search–out, data input/output between the CNC and the floppy, or file deletion.
Floppy 1



Floppy 2



Since floppy replacement is processed by the input/output device, no special operation is required. The CNC will interrupt data input/output operation until the next floppy is inserted into the adaptor.
When reset operation is applied to the CNC during a request for floppy replacement, the CNC is not reset at once, but reset after the floppy has been replaced.

• Protect switch

The floppy is provided with the write protect switch. Set the switch to the write enable state. Then, start output operation.

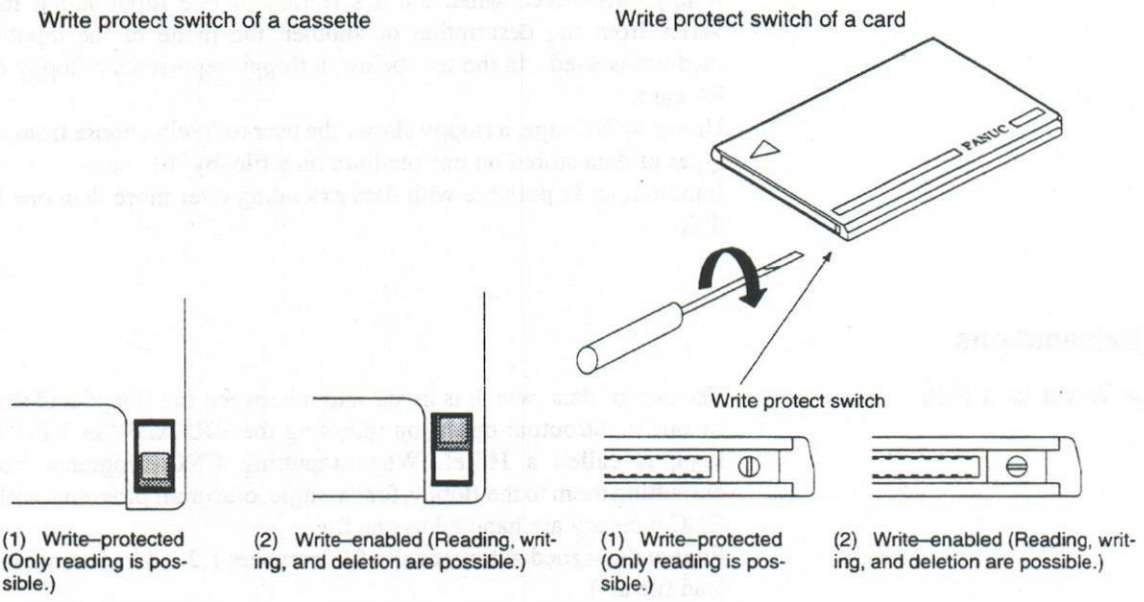

Write protect switch of a cassette
Write protect switch of a card
FANUC
Write protect switch
(1) Write–protected (Only reading is possible.)
(2) Write–enabled (Reading, writing, and deletion are possible.)
(1) Write–protected (Only reading is possible.)
(2) Write–enabled (Reading, writing, and deletion are possible.)

Fig. 8.1 Protect switch

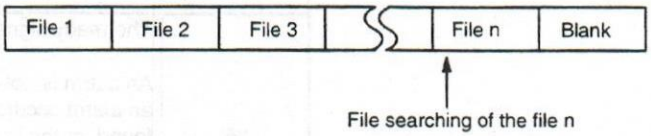
• Writing Memo

Once written in the cassette or card, data can subsequently be read out by correspondence between the data contents and file numbers. This correspondence cannot be verified, unless the data contents and file numbers are output to the CNC and displayed.
To display the contents,write the file numbers and the contents on the memo column which is the back of floppy.

(Entry example on MEMO)
File 1 NC parameters
File 2 Offset data
File 3 NC program O0100
. .
. .
. .
File (n–1) NC program O0500
File n NC program O0600

8.2 FILE SEARCH

When the program is input from the floppy, the file to be input first must be searched.
For this purpose, proceed as follows:



Procedure for File Heading

1 Press the EDIT switch on the machine operator's panel.

2 Press function key [SYSTEM].

3 Press the rightmost soft key [▷] twice (next–menu key).

4 Press soft key **[ALL IO]**

5 Press soft key **[(OPRT)]**

6 Enter address N.

7 Enter the number of the file to search for.
·N0
The beginning of the cassette or card is searched.
·One of N1 to N9999
Of the file Nos. 1 to 9999, a designated file is searched.
·N–9999
The file next to that accessed just before is searched.
·N–9998
When N–9998 is designated, N–9999 is automatically inserted each time a file is input or output. This condition is reset by the designation of N1,N1 to 9999, or N – 9999 or reset.

8 Press soft keys **[FSRH]** and **[EXEC]**
The specified file is searched for.

Explanation

- **File search by N–9999**

The same result is obtained both by sequentially searching the files by specifying Nos. N1 to N9999 and by first searching one of N1 to N9999 and then using the N–9999 searching method. The searching time is shorter in the latter case.

Alarm

No.	Description
86	The ready signal (DR) of an input/output device is off. An alarm is not immediately indicated in the CNC even when an alarm occurs during head searching (when a file is not found, or the like). An alarm is given when the input/output operation is performed after that. This alarm is also raised when N1 is specified for writing data to an empty floppy. (In this case, specify N0.)

8.3 FILE DELETION

Files stored on a floppy can be deleted file by file as required.

Procedure for File Deletion

1 Insert the floppy into the input/output device so that it is ready for writing.

2 Press the EDIT switch on the machine operator's panel.

3 Press function key [SYSTEM] .

4 Press the rightmost soft key [▷] twice (next–menu key).

5 Press soft key **[ALL IO]**

6 Press soft key **[(OPRT)]**

7 Enter address N.

8 Enter the number (from 1 to 9999) of the file to delete.

9 Press soft key **[DELETE]** and then press **[EXEC]** key.
The file specified in step 7 is deleted.

Explanations

- **File number after the file is deleted**

When a file is deleted, the file numbers after the deleted file are each decremented by one. Suppose that a file numbered k was deleted. In this case, files are renumbered as follows:

Before deletion	after deletion
1A(k>1)	1A(k>1)
k Deleted	
(k+1)An	kA(n>1)

- **Protect switch**

Set the write protect switch to the write enable state to delete the files.

8.4 PROGRAM INPUT/OUTPUT

8.4.1 Inputting a Program

This section describes how to load a program into the CNC memory from a floppy or NC tape.

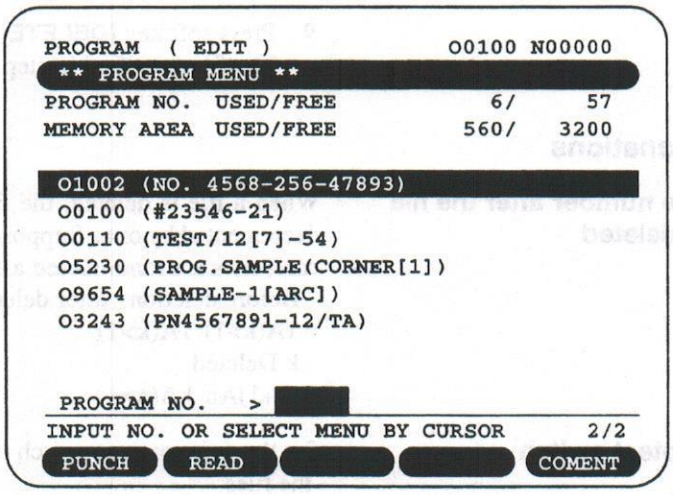
Procedure for Inputting a Program

1 Make sure the input device is ready for reading.

2 Press the EDIT switch on the machine operator's panel.

3 Press function key [PROG] . Then the program list screen is displayed.

4 Press the rightmost soft key [▷] (next–menu key). Then the soft key **[READ]** is displayed.



5 Press the **[READ]** soft key. Programs are read and registered into memory.
Before pressing the **[READ]** soft key, check that the data protection key on the machine operator's panel is released.
When a floppy disk is used, only the first file is read.

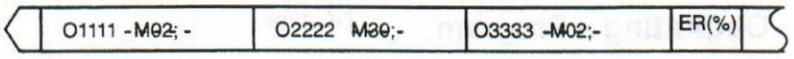
Explanations

- **Collation**

If a program is input while the data protect key on the machine operator's panel turns ON, the program loaded into the memory is verified against the contents of the floppy or NC tape.
If a mismatch is found during collation, the collation is terminated with an alarm (No. 079).
If the operation above is performed with the data protection key turns OFF, collation is not performed, but programs are registered in memory.

- **Inputting multiple programs from an NC tape**

When a tape holds multiple programs, the tape is read up to ER (or %).



- **Program registration in the background**

The method of registration operation is the same as the method of foreground operation. However, this operation registers a program in the background editing area. As with edit operation, the operations described below are required at the end to register a program in foreground program memory.

Program list screen is displayed.

[BG–END]

- **Registering a program having the same number as that of a previously registered program**

The following parameter can be used to specify whether any program having the same number as that of a registered program is to be registered:

Bit 2 (REP) of parameter No. 3201

0 : The program is not registered, causing P/S alarm 073.

1 : The program is registered after deletion of the already registered program. If editing is disabled, however, an alarm is issued without deleting the registered program.

Alarm

No.	Description
70	The size of memory is not sufficient to store the input programs
73	An attempt was made to store a program with an existing program number.
79	The verification operation found a mismatch between a program loaded into memory and the contents of the program on the floppy or NC tape.

8.4.2 Outputting a Program

A program stored in the memory of the CNC unit is output to a floppy or NC tape.

Procedure for Outputting a Program

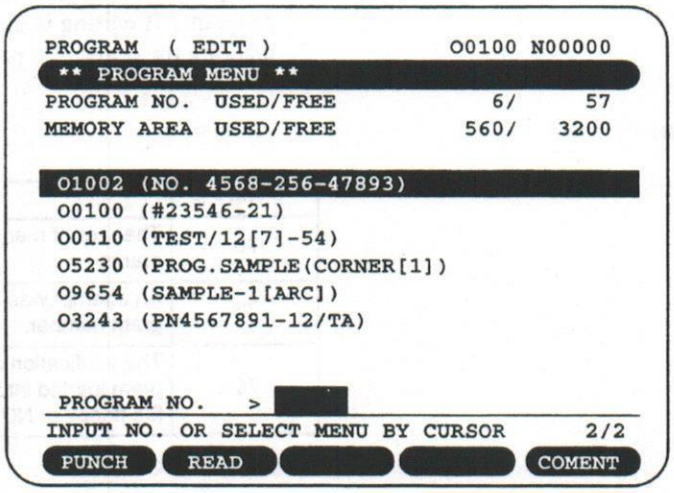
1 Make sure the output device is ready for output.

2 To output to an NC tape, specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key [PROG]. Then the program list screen is displayed.

5 Press the rightmost soft key [▷] (next–menu key). Then soft key **[PUNCH]** is displayed.



6 To output a single program, enter the number of the program by using the numeric keys. Pressing the **[PUNCH]** soft key outputs the program.
To output all programs, press the **[PUNCH]** soft key without entering a program number. All programs displayed on the program list screen are output.

Explanations (Output to a floppy)

- **File output location**

When output is conducted to the floppy, the program is output as the new file after the files existing in the floppy. New files are to be written from the beginning with making the old files invalid, use the above output operation after the N0 head searching.

- **An alarm while a program is output**

When the alarm (No.086) occurs during program output, the floppy is restored to the condition before the output.

- **Outputting a program after file heading**

When program output is conducted after N1 to N9999 head searching, the new file is output as the designated n–th position. In this case, 1 to n–1 files are effective, but the files after the old n–th one are deleted. If an alarm occurs during output, only the 1 to n–1 files are restored.

- **Efficient use of memory**

To efficiently use the memory in the cassette or card, output the program by setting parameter NFD (No. 0101#7,No. 0111#7 or 0121#7) to 1. This parameter makes the feed is not output, utilizing the memory efficiently.

- **On the memo record**

Head searching with a file No. is necessary when a file output from the CNC to the floppy is again input to the CNC memory or compared with the content of the CNC memory. Therefore, immediately after a file is output from the CNC to the floppy, record the file No. on the memo.

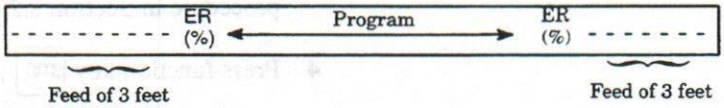
- **Punching programs in the background**

Punch operation can be performed in the same way as in the foreground. This function alone can punch out a program selected for foreground operation.
The program list screen is displayed.
Program No. **[PUNCH]**: Punches out a specified program.
[PUNCH]: Punches out all programs.

Explanations (Output to an NC tape)

- **Format**

A program is output to paper tape in the following format:



Bit 3 (NCR) of parameter No.0100 can be set to display LF only.

If three–feet feeding is too long, press the CAN key during feed punching to cancel the subsequent feed punching.

- **TV check**

A space code for TV check is automatically punched.

- **ISO code**

When a program is punched in ISO code, two CR codes are punched after an LF code.

```
- - - - - - - - - - - - - - LF CR CR
```

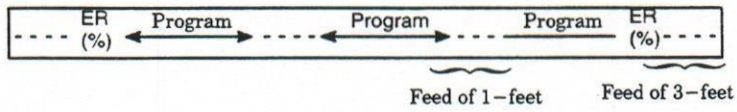
Bit 3 (NCR) of parameter No. 0100 can be set to display LF only.

- **Stopping the punch**

Press the RESET key to stop punch operation.

- **Punching all programs**

All programs are output to paper tape in the following format.



The sequence of the programs punched is undefined.

8.5 OFFSET DATA INPUT AND OUTPUT

8.5.1 Inputting Offset Data

Offset data is loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as for offset value output. See section **8.5.2.** When an offset value is loaded which has the same offset number as an offset number already registered in the memory, the loaded offset data replaces existing data.

Procedure for Inputting Offset Data

1 Make sure the input device is ready for reading

2 Press the EDIT switch on the machine operator's panel.

3 When using a floppy, search for the required file according to the procedure in Section 8.2.

4 Press function key [OFFSET SETTING], and then the offset screen is displayed.

5 Press soft keys **[(OPRT)]**.

6 Press rightmost soft key [▷] (next menu key).

7 Press soft keys **[READ]** and **[EXEC]**.

8 The input offset data will be displayed on the screen after completion of input operation.

8.5.2 Outputting Offset Data

All offset data is output in a output format from the memory of the CNC to a floppy or NC tape.

Procedure for Outputting Offset Data

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key [OFFSET SETTING], and then the offset screen is displayed.

5 Press soft key **[(OPRT)]**.

6 Press the rightmost soft key [▷] (next–menu key)

7 Press soft keys **[PUNCH]** and **[EXEC]**.
Offset data is output in the output format described below.

Explanations

- **Output format**

Output format is as follows:

Format

G10P_X_Z_R_Q_;
P: Offset number
. . . Work sheet
:P=0
. . . For wear offset amount
:P=Wear offset number
. . . For geometry offset amount
:p=10000+geometry offset number
X:Offset value on X axis
Z:Offset value on Z axis
Q:Imaginary tool nose number
R:Tool nose radius offset value

The L1 command may be used instead of L11 for format compatibility of the conventional CNC.

8.6 INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA

Parameters and pitch error compensation data are input and output from different screens, respectively. This chapter describes how to enter them.

8.6.1 Inputting Parameters

Parameters are loaded into the memory of the CNC unit from a floppy or NC tape. The input format is the same as the output format. See Section 8.6.2. When a parameter is loaded which has the same data number as a parameter already registered in the memory, the loaded parameter replaces the existing parameter.

Procedure for Inputting Parameters

1 Make sure the input device is ready for reading.

2 When using a floppy, search for the required file according to the procedure in Section 8.2.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key [OFFSET SETTING].

5 Press the soft key **[SETING]** for chapter selection.

6 Enter 1 to "PARAMETER WRITE" in the setting data (PWE).
Then alarm (No.100) (indicating that parameters can be written) appears.

7 Press soft key [SYSTEM].

8 Press chapter selection soft key **[PARAM]** to display the parameter screen.

9 Press soft key **[(OPRT)]**.

10 Press the rightmost soft key [▷] (next–menu key).

11 Press soft keys **[READ]** and **[EXEC]**.
Parameters are read into memory. Upon completion of input, the "INPUT" indicator at the lower–right corner of the screen disappears.

12 Press function key [OFFSET SETTING].

13 Press soft key **[SETING]** for chapter selection.

14 Enter 0 to "PARAMETER WRITE" in the setting data.

15 Turn the power to the NC back on.

16 Release the EMERGENCY STOP button on the machine operator's panel.

8.6.2 Outputting Parameters

All parameters are output in the defined format from the memory of the CNC to a floppy or NC tape.

Procedure for Outputting Parameters

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key [SYSTEM].

5 Press chapter selection soft key **[PARAM]** to display the parameter screen.

6 Press soft key **[(OPRT)]**.

7 Press rightmost soft key [▷] (next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
All parameters are output in the defined format.

Explanations

- **Output format**

Output format is as follows:
NP ;
NA1P A2P AnP ;
NP ;

N:Parameter No.
A:Axis No.(n is the number of control axis)
P:Parameter setting value .

8.6.3 Inputting Pitch Error Compensation Data

Pitch error compensation data are loaded into the memory of the CNC from a floppy or NC tape. The input format is the same as the output format. See Section **8.6.4.** When a pitch error compensation data is loaded which has the corresponding data number as a pitch error compensation data already registered in the memory, the loaded data replaces the existing data.

Procedure for Pitch Error Compensation Data

1 Make sure the input device is ready for reading.

2 When using a floppy, search for the required file according to the procedure in Section **8.2**.

3 Press the EMERGENCY STOP button on the machine operator's panel.

4 Press function key [OFFSET SETTING].

5 Press the soft key **[SETING]** for chapter selection.

6 Enter 1 to "PARAMETER WRITE" in the setting data (PWE). Then alarm (No.100) (indicating that parameters can be written) appears.

7 Press soft key [SYSTEM].

8 Press the rightmost soft key [▷] (next–menu key)and press chapter selection soft key **[PITCH]**.

9 Press soft key **[(OPRT)]**.

10 Press the rightmost soft key [▷] (next–menu key).

11 Press soft keys **[READ]** and **[EXEC]**.
Pitch error compensation data are read into memory. Upon completion of input, the "INPUT" indicator at the lower–right corner of the screen disappears.

12 Press function key [OFFSET SETTING].

13 Press soft key **[SETING]** for chapter selection.

14 Enter 0 to "PARAMETER WRITE" in the setting data.

15 Turn the power to the CNC back on.

16 Release the EMERGENCY STOP button on the machine operator's panel.

Explanations

- **Pitch error compensation**

Parameters 3620 to 3624 and pitch error compensation data must be set correctly to apply pitch error compensation correctly
(See subsection **11.5.2**)

8.6.4 Outputting Pitch Error Compensation Data

All pitch error compensation data are output in the defined format from the memory of the CNC to a floppy or NC tape.

Procedure for Outputting Pitch Error Compensation Data

1 Make sure the output device is ready for output.

2 Specify the punch code system (ISO or EIA) using a parameter.

3 Press the EDIT switch on the machine operator's panel.

4 Press function key [SYSTEM] .

5 Press the rightmost soft key [▷] (next–menu key) and press chapter selection soft key **[PITCH]**.

6 Press soft key **[(OPRT)]**.

7 Press rightmost soft key [▷] (next–menu key).

8 Press soft keys **[PUNCH]** and **[EXEC]**.
All pitch error compensation data are output in the defined format.

Explanations

- **Output format**

Output format is as follows:
N 10000 P ;
N 11023 P ;
N:Pitch error compensation point No. +10000
P:Pitch error compensation data

8.7 INPUTTING/ OUTPUTTING CUSTOM MACRO COMMON VARIABLES

8.7.1 Inputting Custom Macro Common Variables

The value of a custom macro common variable (#500 to #531) is loaded into the memory of the CNC from a floppy or NC tape. The same format used to output custom macro common variables is used for input. See Section **8.7.2**. For a custom macro common variable to be valid, the input data must be executed by pressing the cycle start button after data is input. When the value of a common variable is loaded into memory, this value replaces the value of the same common variable already existing (if any) in memory.

Procedure for Inputting Custom Macro Common Variables

1 Register the program output in Section **8.7.2** in memory, according to the procedure given in Section **8.4.1**.

2 Press the MEMORY switch on the machine operator's panel upon completing input.

3 Press the cycle start button to execute the loaded program.

4 Display the macro variable screen to check whether the values of the common variables have been set correctly.

Display of the macro variable screen

·Press function key [OFFSET SETTING] .
·Press the rightmost soft key (next–menu key).
·Press soft key **[MACRO]**.
·Select a variable with the page keys or numeric keys and soft key **[NO.SRH]**.

Explanations

- **Common variables**

The common variables (#500 to #531) can be input and output. Variables #100 to #149 can also be input and output when bit 3 (PV5) of parameter No.6001 is set to 1.

8.7.2 Outputting Custom Macro Common Variable

Custom macro common variables (#500 to #531) stored in the memory of the CNC can be output in the defined format to a floppy or NC tape.

Procedure for Outputting Custom Macro Common Variable

1 Make sure the output device is ready for output.
2 Specify the punch code system (ISO or EIA) using a parameter.
3 Press the EDIT switch on the machine operator's panel.
4 Press function key [OFFSET SETTING].
5 Press the rightmost soft key [▷] (next–menu key), then press soft key **[MACRO]**.
6 Press soft key **[(OPRT)]**.
7 Press the rightmost soft key [▷] (next–menu key).
8 Press soft keys **[PUNCH]** and **[EXEC]**.
Common variables are output in the defined format.

Explanations

- **Output format**

The output format is as follows:

```
%
;
#500=[25283*65536+65536]/134217728.....
(1)
#501=#0; (2)
#502=0;  (3)
#503=  .  ;
.......  ;
.......  ;
#531=  .  ;
M02;
%
```

(1) The precision of a variable is maintained by outputting the value of the variable as <expression>.
(2) Undefined variable
(3) When the value of a variable is 0

- **Common variable**

The common variables (#500 to #531) can be input and output.
Variables #100 to #149 can also be input and output when bit 3 (PV5) of parameter No.6001 is set to 1.

9 EDITING PROGRAMS

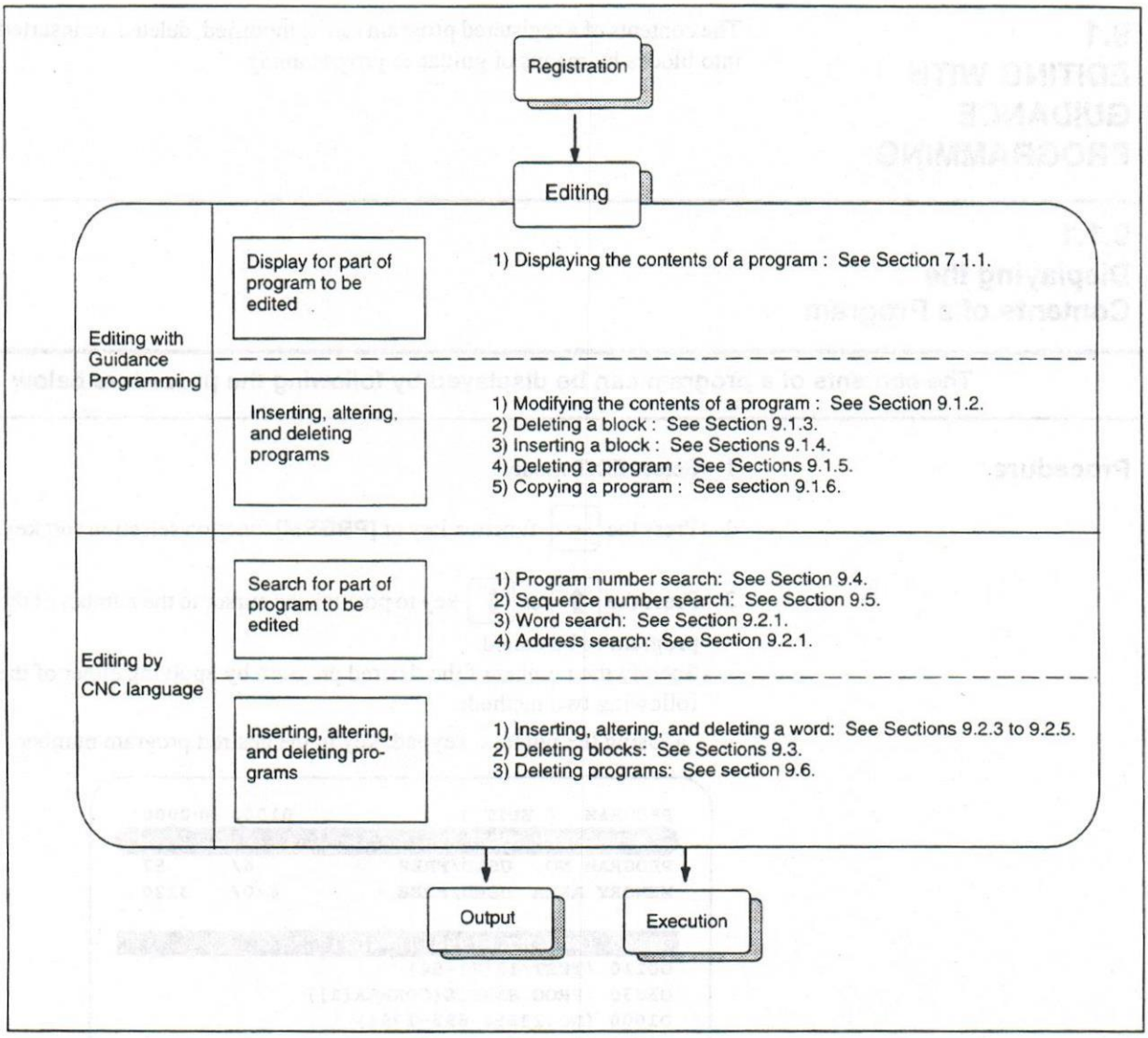
This chapter describes how to edit a program registered in the CNC. Programs can be edited using either of the following two methods:

- Editing by guidance programming
- Editing by the CNC language

1. Editing by guidance programming enables the modification of a program and the deletion and insertion of a block according to guidance, as well as the deletion and copying of the entire program.

2. Editing by the CNC language enables the insertion, modification, and deletion of a word, as well as the deletion of the entire program. This chapter also describes searching for a program number, sequence number, word, or address, prior to program editing.

The program editing function can be used when the guidance programming function option is supported.


Registration
Editing
Editing with Guidance Programming
Display for part of program to be edited
1) Displaying the contents of a program : See Section 7.1.1.
Inserting, altering, and deleting programs
1) Modifying the contents of a program : See Section 9.1.2.
2) Deleting a block : See Section 9.1.3.
3) Inserting a block : See Sections 9.1.4.
4) Deleting a program : See Sections 9.1.5.
5) Copying a program : See section 9.1.6.
Editing by CNC language
Search for part of program to be edited
1) Program number search: See Section 9.4.
2) Sequence number search: See Section 9.5.
3) Word search: See Section 9.2.1.
4) Address search: See Section 9.2.1.
Inserting, altering, and deleting programs
1) Inserting, altering, and deleting a word: See Sections 9.2.3 to 9.2.5.
2) Deleting blocks: See Sections 9.3.
3) Deleting programs: See section 9.6.
Output
Execution

9.1 EDITING WITH GUIDANCE PROGRAMMING

The contents of a registered program can be modified, deleted, or inserted into blocks by means of guidance programming.

9.1.1 Displaying the Contents of a Program

The contents of a program can be displayed by following the procedure below

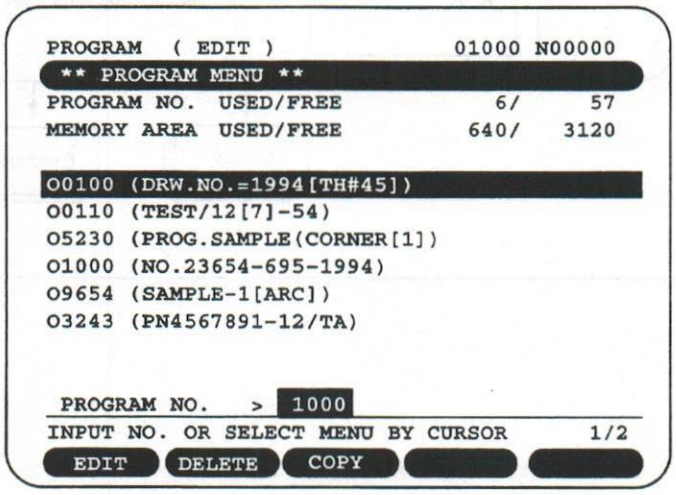
Procedure

1 Select **EDIT** mode.

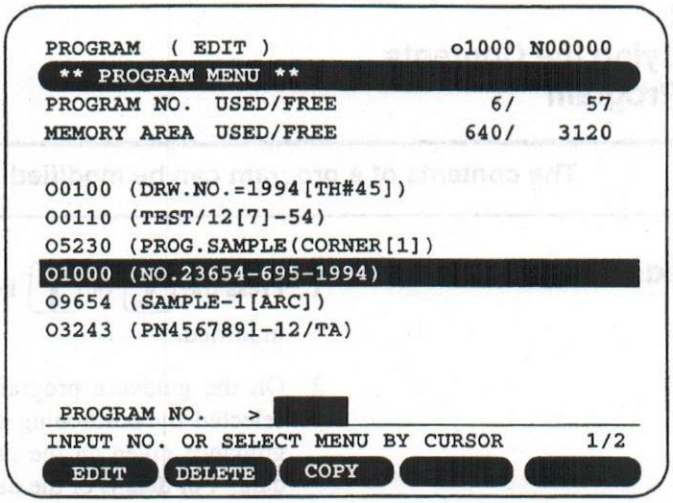
2 Press the [PROG] function key or **[PRGRM]** function selection soft key.

3 Press the [↑] or [↓] key to position the cursor to the number of the program to be edited.
Specify the number of the desired program by applying either of the following two methods:

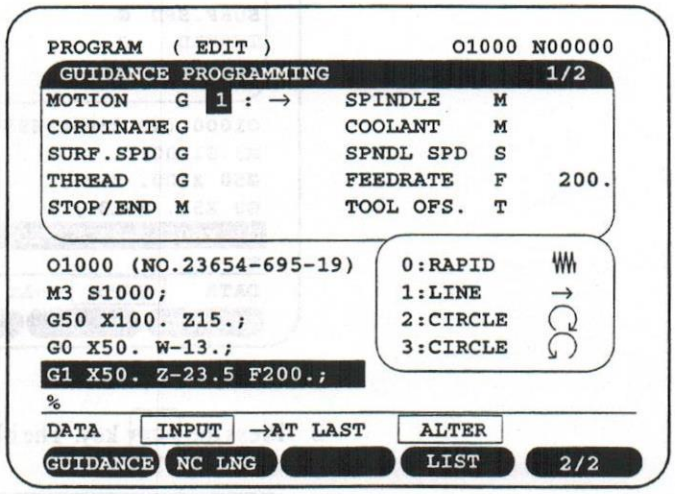
a. Using the numeric keypad, key in the desired program number.



b.Position the cursor to the desired program number.



4 Press the **[EDIT]** soft key.
The contents of the block selected by positioning the cursor are displayed on the guidance programming screen.



5 To move the displayed block, press page key or .

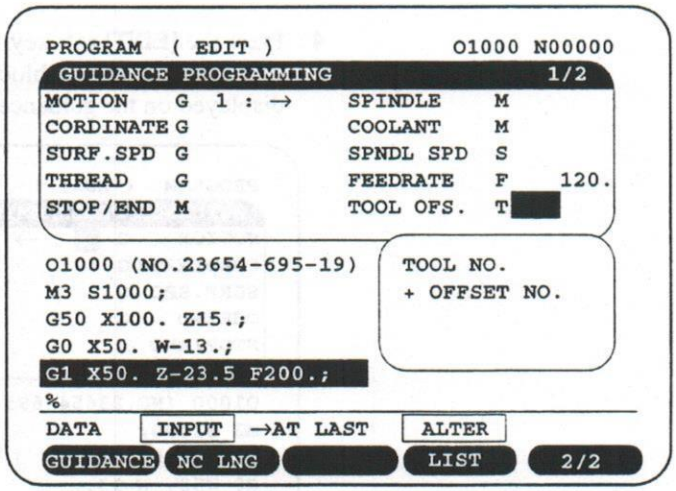
9.1.2 Modifying the Contents of a Program

The contents of a program can be modified by following the procedure below

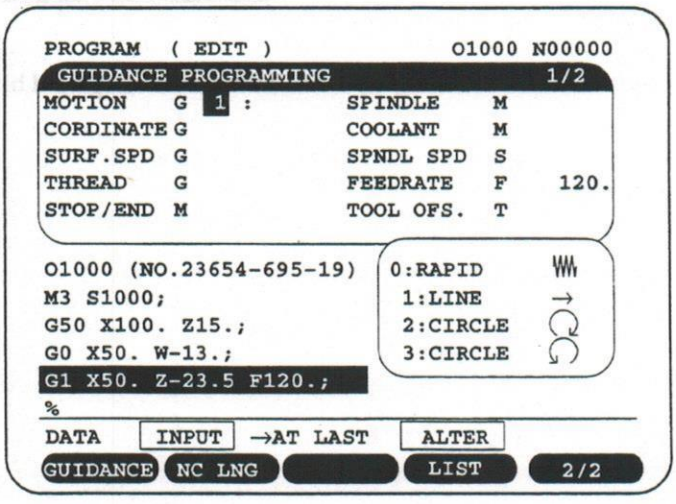
Procedure

1 Press the [PAGE ↑] or [PAGE ↓] key to position the cursor to the block to be modified.

2 On the guidance programming screen, the contents of the block selected by positioning the cursor are displayed. Following the guidance given on the guidance programming screen, modify the data. For details of the data input method, see Section 10.1.4.



3 Press the [ALTER] key. The block is modified according to the input data.

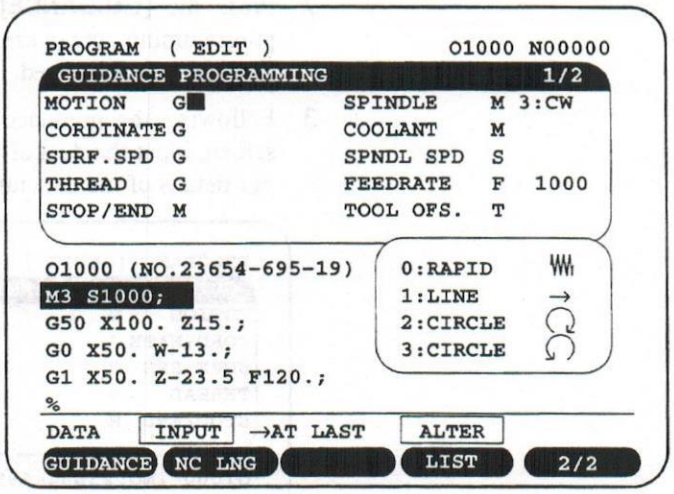


9.1.3 Deleting a Block

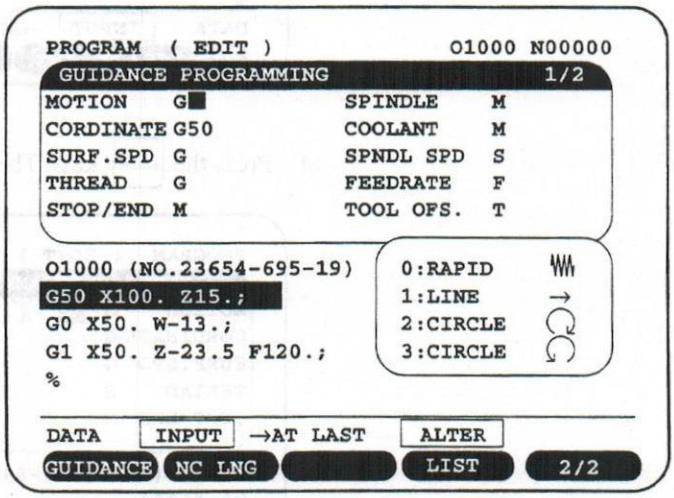
A block is deleted by following the procedure below

Procedure

1 Press the [PAGE ↑] or [PAGE ↓] key to position the cursor to the block to be deleted.



2 Press the [DELETE] key. The selected block is deleted.



9.1.4 Inserting a Block

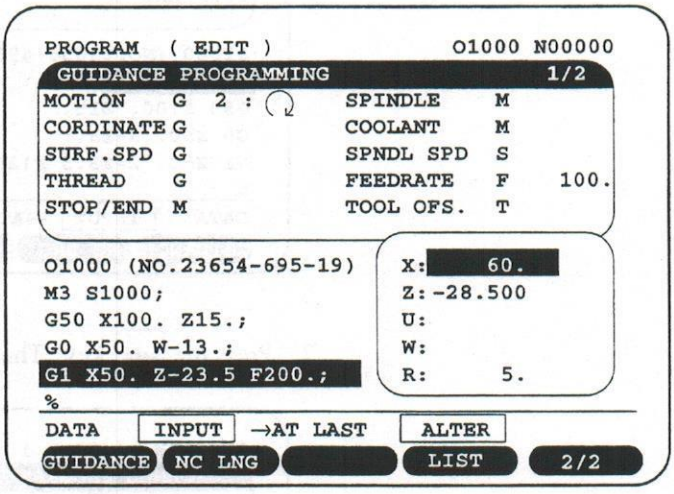
A block is inserted by following the procedure below

Procedure

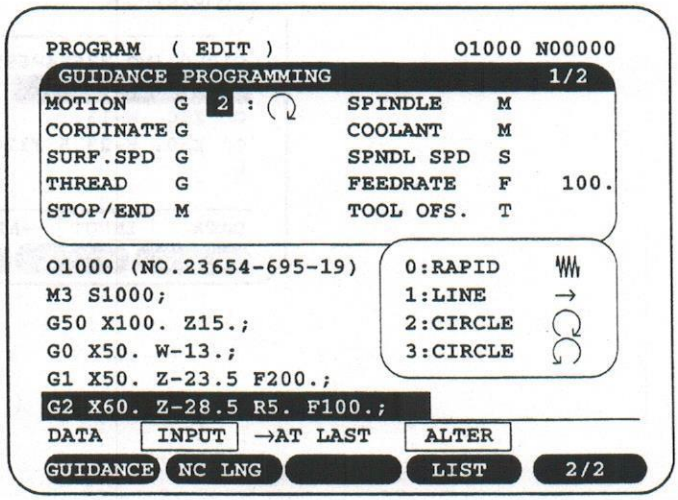
1 Press the [PAGE ↑] or [PAGE ↓] key to position the cursor to the last block prior to the position where the block is to be inserted.

2 Press the **[GUIDANCE]** soft key. The entire contents of the programming screen are initialized and the data of the block to be inserted can be entered.

3 Following the guidance displayed on the guidance programming screen, input the data of the block to be inserted.
For details of the data input method, see Section 10.1.4.



4 Press the [INSERT] key. The block is inserted.



9.1.5 Deleting a Program

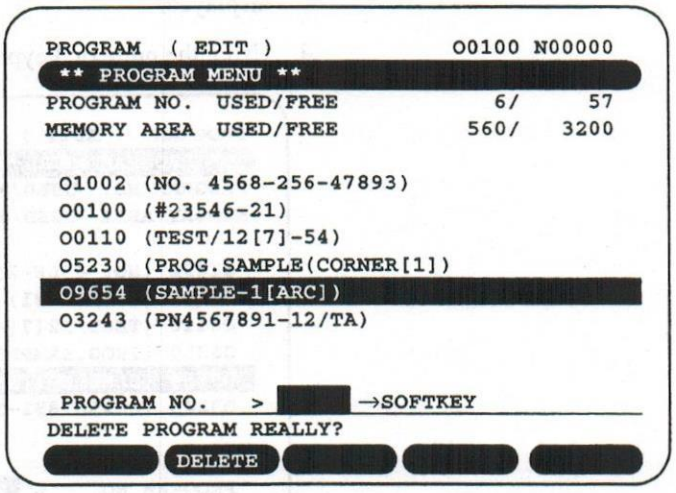
A program is deleted by following the procedure below

Procedure

1 Display the program menu screen.

2 Using the numeric keypad, enter the number of the program to be deleted. Alternatively, position the cursor to the desired program number.

3 Press the **[DELETE]** soft key.
Check that the data protect key on the machine operator's panel is released.
The message "DELETE PROGRAM REALLY?" is displayed.



4 To delete the program, press the **[DELETE]** soft key again. To abandon the deletion, press the leftmost soft key or the RESET key. The screen displayed in 1, above, reappears.

9.1.6 Copying a Program

The program copy function is provided to enable part of a existing program to be modified to create another program.

A program is copied by following the procedure below

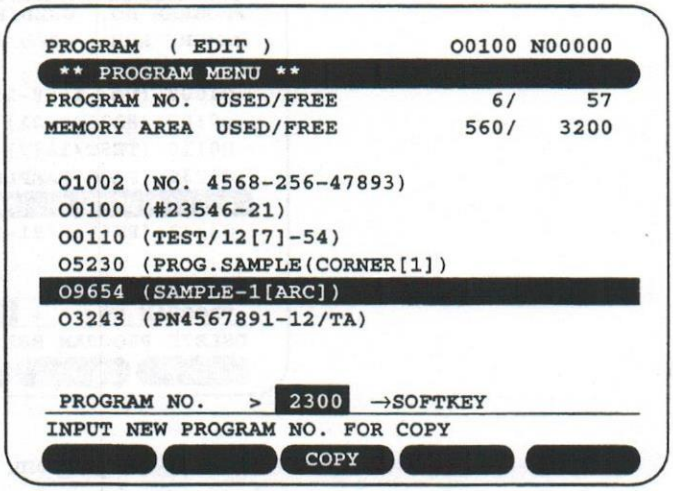
Procedure

1 Press the rightmost soft key. The **[COPY]** soft key is displayed.

2 Using the numeric keypad, enter the number of the program to be copied. Alternatively, position the cursor to the desired program number.

3 Press the **[COPY]** soft key.
Check that the data protect key on the machine operator's panel is released.
The message "INPUT NEW PROGRAM NO. FOR COPY" is displayed.

4 Using the numeric keypad, enter the new program number.



5 Press the **[COPY]** soft key once again. The program is registered with the new program number. If the specified new program number has already been registered using the program menu screen, the message "PROGRAM NUMBER ALREADY EXISTS" is displayed. The program cannot be copied.

9.1.7 Renaming a Program

A program name, registered using the program menu screen, can be changed.

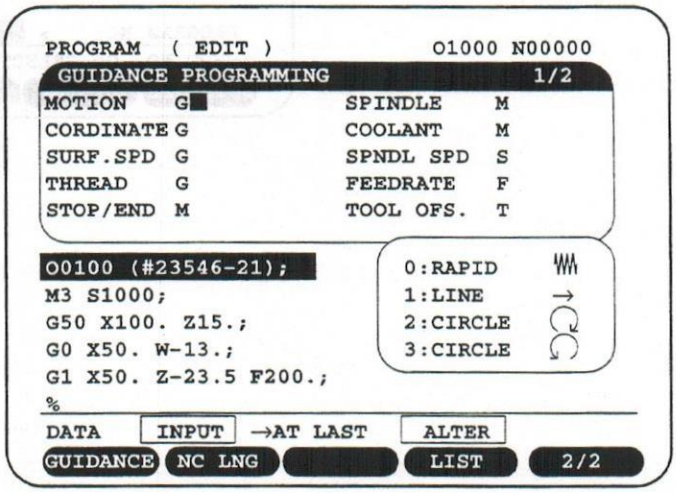
A program is renamed by following the procedure below

Procedure

1 Display the program menu screen.

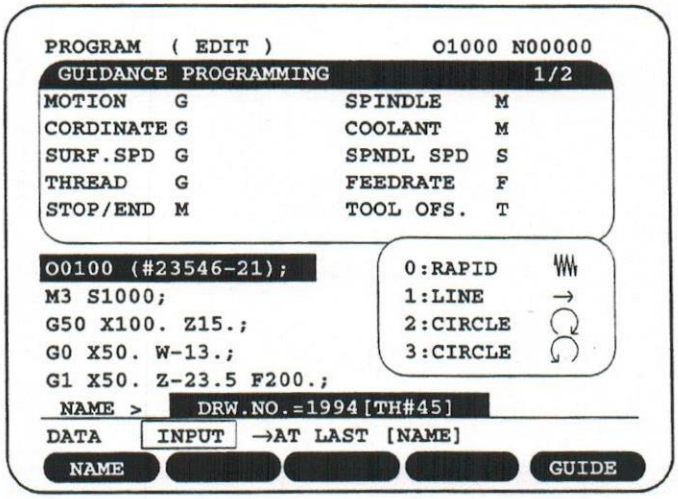
2 Using the numeric keypad, enter the number of the program to be renamed. Alternatively, position the cursor to the desired program number.

3 Press the **[EDIT]** soft key.

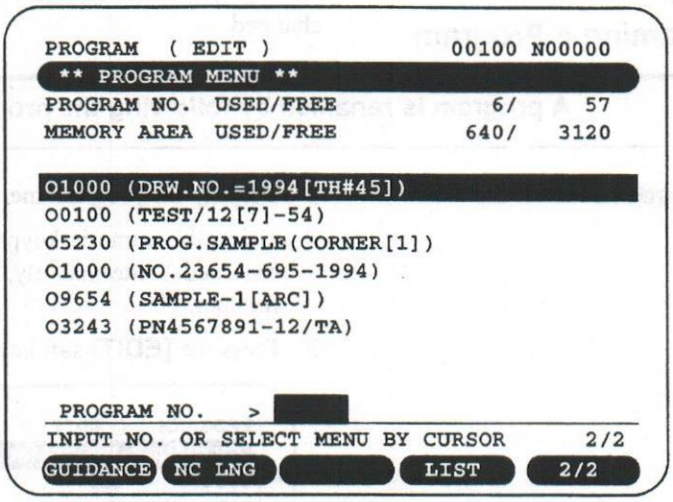


4 Press the rightmost soft key. The **[NAME]** soft key is displayed. Note that the **[NAME]** soft key is displayed only when the data protect key is released.

5 Using the address and numeric keys, enter a new program name of up to 23 characters.
Do not include the EOB (;) in a program name.



6 Press the **[NAME]** soft key. The program is renamed.



9.1.8 Editing in CNC Language

When the **[NC LNG]** soft key is pressed on the guidance programming screen, the CNC language editing screen is displayed. Using this screen, programs can be edited in CNC language.

Editing in the CNC language is done by following the procedure below

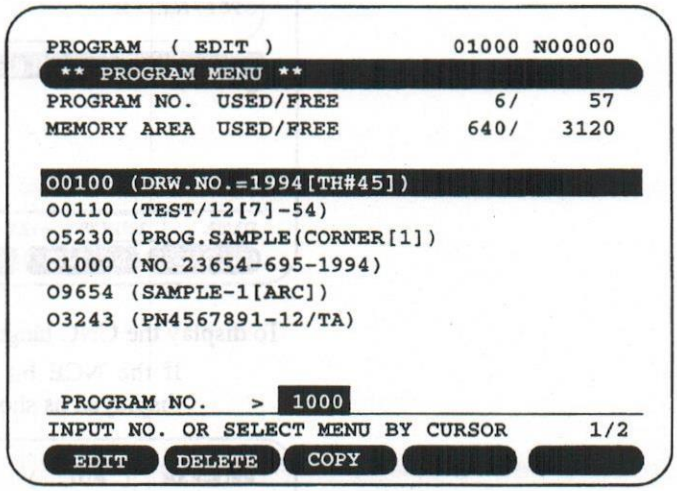
Procedure

1 Select **EDIT** mode.

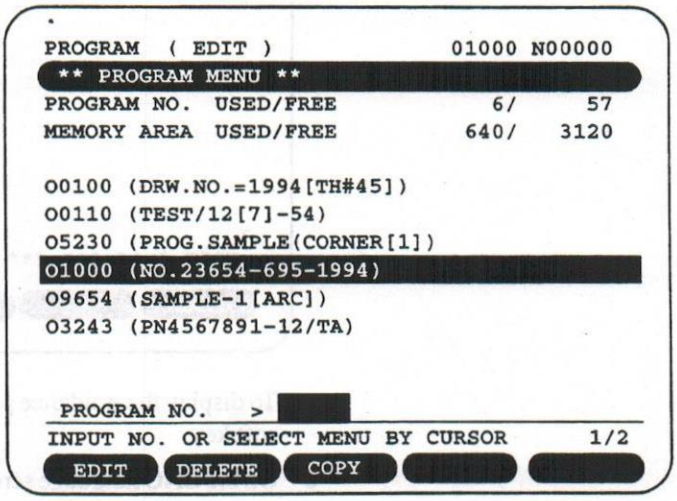
2 Press the PROG function key or the **[PRGRM]** soft key.

3 Press the ↑ or ↓ key to position the cursor to the number of the program to be edited.
Specify the desired program number by means of either of the following two methods:

a. Using the numeric keypad, enter the desired program number.



b. Position the cursor to the desired program number.



4 Press the **[EDIT]** soft key.

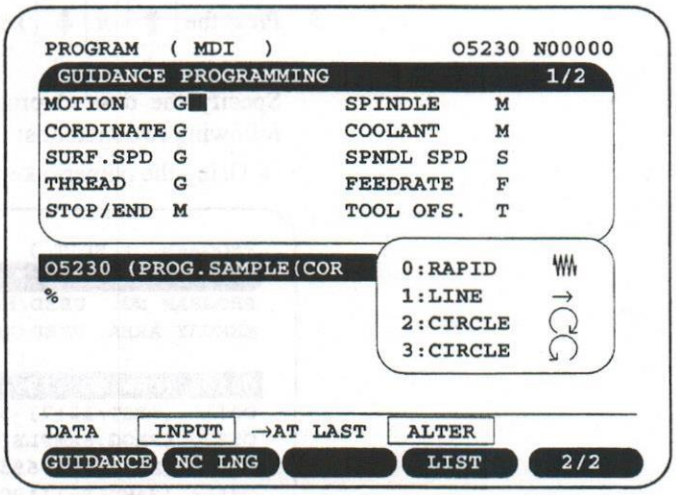
5 The guidance programming screen or CNC language screen is displayed. The screen displayed depends on the following this parameter:

Bit 0 of Parameter 9320 (NCE)

0: Displays the guidance programming screen after a program number is registered.

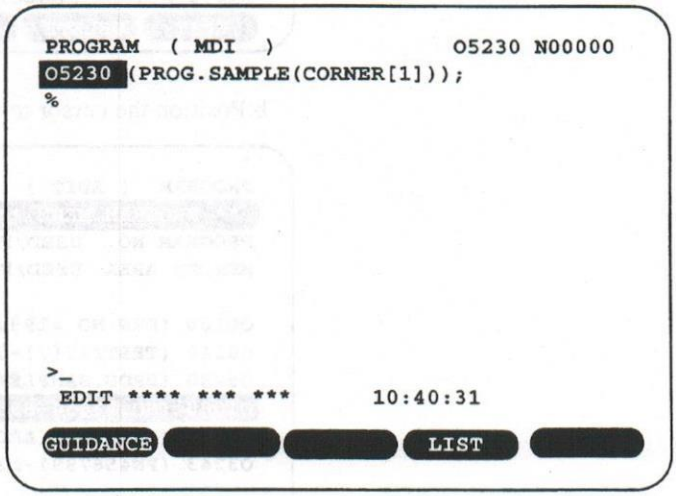
1: Displays the CNC language screen after a program number is registered.

If the NCE bit is set to 0, the guidance programming screen is displayed as shown below:



To display the CNC language screen, press the **[NC LNG]** soft key.

If the NCE bit is set to 1, the CNC language screen is displayed as shown below:



To display the guidance programming screen, press the **[GUIDANCE]** soft key.

6 On the CNC language screen, edit a program, as explained in Sections 9.2 to 9.7.

9.2 INSERTING, ALTERING AND DELETING A WORD

This section outlines the procedure for inserting, modifying, and deleting a word in a program registered in memory.

Procedure for inserting, altering and deleting a word

1 Select **EDIT** mode.

2 Display the CNC language screen accordind to the procedure for editing by the CNC language, as described in Section **9.1.8**.

3 Search for a word to be modified.
·Scan method
·Word search method

4 Perform an operation such as altering, inserting, or deleting a word.

Explanation

- **Concept of word and editing unit**

A word is an address followed by a number. With a custom macro, the concept of word is ambiguous.
So the editing unit is considered here.
The editing unit is a unit subject to alteration or deletion in one operation.
In one scan operation, the cursor indicates the start of an editing unit.
An insertion is made after an editing unit.
Definition of editing unit
(i) Program portion from an address to immediately before the next address
(ii) An address is an alphabet, **IF**, **WHILE**, **GOTO**, **END**, **DO=**,or ; **(EOB)**.
According to this definition, a word is an editing unit.
The word "word," when used in the description of editing, means an editing unit according to the precise definition.

Notes

The user cannot continue program execution after altering, inserting, or deleting data of the program by suspending machining in progress by means of an operation such as a single block stop or feed hold operation during program execution. If such a modification is made, the program may not be executed exactly according to the contents of the program displayed on the screen after machining is resumed. So, when the contents of memory are to be modified by part program editing, be sure to enter the reset state or reset the system upon completion of editing before executing the program.

9.2.1 Word Search

A word can be searched for by merely moving the cursor through the text (scanning), by word search, or by address search.

Procedure for scanning a program

1 Press the cursor key [→]

The cursor moves forward word by word on the screen; the cursor is displayed at a selected word.

2 Press the cursor key [←]

The cursor moves backward word by word on the screen; the cursor is displayed at a selected word.

Example) When Z1250.0 is scanned

```
Program                          O0050 N01234
O0050 ;
N01234 X100.0 ■■■■■■ ;
S12 ;
N56789 M03 ;
M02 ;
%
```

3 Holding down the cursor key [→] or [←] scans words continuously.

4 The first word of the next block is searched for when the cursor key [↓] is pressed.

5 The first word of the previous block is searched for when the cursor key [↑] is pressed.

6 Holding down the cursor key [↓] or [↑] moves the cursor to the head of a block continuously.

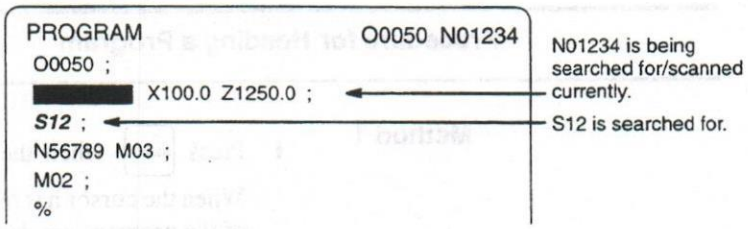
7 Pressing the page key [PAGE ↓] displays the next page and searches for the first word of the page.

8 Pressing the page key [↑ PAGE] displays the previous page and searches for the first word of the page.

9 Holding down the page key [PAGE ↓] or [↑ PAGE] displays one page after another.

Procedure for searching a word (valid for only MDI mode)

Example) of Searching for S12



1 Key in address [S].

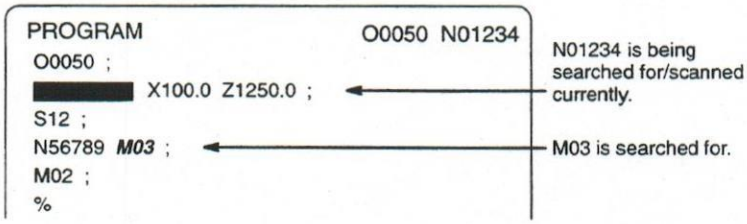
2 Key in [1] [2].

·S12 cannot be searched for if only S1 is keyed in.
·S09 cannot be searched for by keying in only S9.
To search for S09, be sure to key in S09.

3 Pressing the **[SRH↓]** key starts search operation.
Upon completion of search operation, the cursor is displayed at S12.
Pressing the **[SRH↑]** key rather than the **[SRH↓]** key performs search operation in the reverse direction.

Procedure for searching an address (valid for only MDI mode)

Example) of Searching for M03



1 Key in address [M].

2 Press the **[SRH↓]** key.
Upon completion of search operation, the cursor is displayed at M03.
Pressing the **[SRH↑]** key rather than the **[SRH↓]** key performs search operation in the reverse direction.

Alarm

Alarm number	Description
71	The word or address being searched for was not found.

9.2.2 Heading a Program

The cursor can be jumped to the top of a program. This function is called heading the program pointer. This section describes the three methods for heading the program pointer.

Procedure for Heading a Program

Method 1

1 Press [RESET] when the program screen is selected in EDIT mode. When the cursor has returned to the start of the program, the contents of the program are displayed from its start on the screen.

Method 2

Search for the program number.

1 Press address [O] ,when a program screen is selected in the **MEMORY** mode.

2 Input a program number.

3 Press the soft key **[O SRH]**.

Method 3

1 Select **MEMORY** mode.

2 Press [PROG] .

3 Press the **[(OPRT)]** key.

4 Press the **[REWIND]** key.

9.2.3 Inserting a Word

Procedure for inserting a word

1 Search for or scan the word immediately before a word to be inserted.
2 Key in an address to be inserted.
3 Key in data.
4 Press the INSERT key.

Example of Inserting T15

Procedure

1 Search for or scan Z1250.

```
Program                              O0050 N01234
O0050 ;
N01234 X100.0 [redacted] ;   <-- Z1250.0 is searched for/scanned.
S12 ;
N56789 M03 ;
M02 ;
%
```

2 Key in T 1 5 .

3 Press the INSERT key.

```
Program                              O0050 N01234
O0050 ;
N01234 X100.0 Z1250.0 T15 ;   <-- T15 is inserted.
S12 ;
N56789 M03 ;
M02 ;
%
```

9.2.4 Altering a Word

Procedure for altering a word

1 Search for or scan a word to be altered.
2 Key in an address to be inserted.
3 Key in data.
4 Press the ALTER key.

Example of changing T15 to M15

Procedure

1 Search for or scan T15.

```
Program                              O0050 N01234
O0050 ;
N01234 X100.0 Z1250.0   T15  ;
S12 ;
N56789 M03 ;
M02 ;
%
```

T15 is searched for/ scanned.

2 Key in M 1 5 .

3 Press the ALTER key.

```
Program                              O0050 N01234
O0050 ;
N1234 X100.0 Z1250.0 M15 ;
S12 ;
N5678 M03 ;
M02 ;
%
```

T15 is changed to M15.

9.2.5 Deleting a Word

Procedure for deleting a word

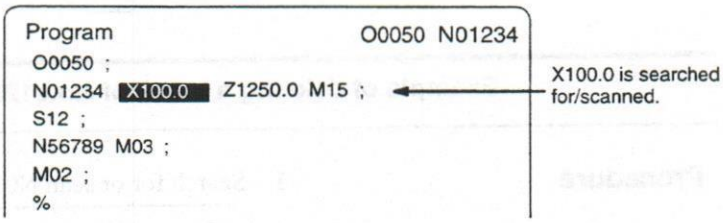
1 Search for or scan a word to be deleted.

2 Press the DELETE key.

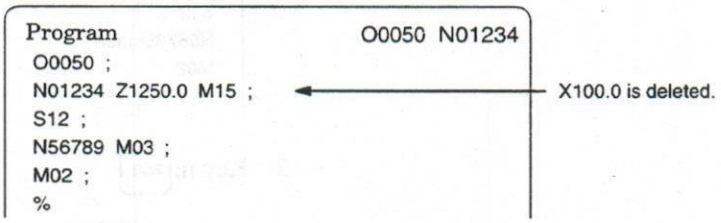
Example of deleting X100.0

Procedure

1 Search for or scan X100.0.



2 Press the DELETE key.



9.3 DELETING BLOCKS

A block or blocks can be deleted in a program.

9.3.1 Deleting a Block

The procedure below deletes a block up to its EOB code; the cursor advances to the address of the next word.

Procedure for deleting a block

1 Search for or scan address N for a block to be deleted.

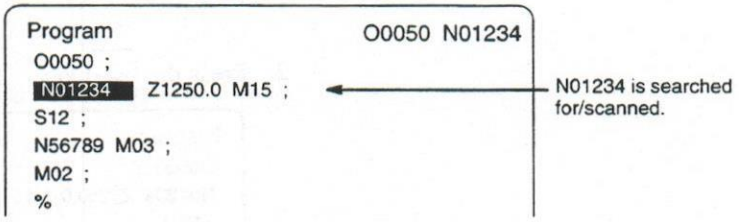
2 Key in [EOB].

3 Press the [DELETE].

Example of deleting a block of No.01234

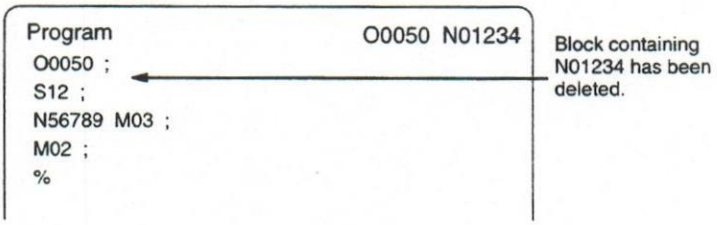
Procedure

1 Search for or scan N01234.



2 Key in [EOB].

3 Press the [DELETE] key.



9.3.2 Deleting Multiple Blocks

The blocks from the currently displayed word to the block with a specified sequence number can be deleted.

Procedure for deleting multiple blocks

1 Search for or scan a word in the first block of a portion to be deleted.

2 Key in address [N] .

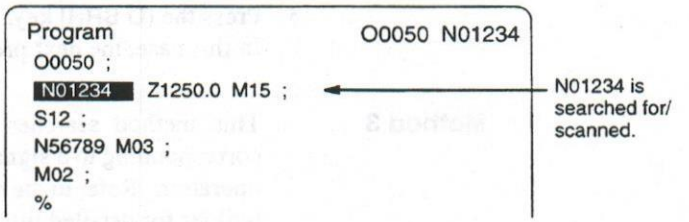
3 Key in the sequence number for the last block of the portion to be deleted.

4 Press the [DELETE] key.

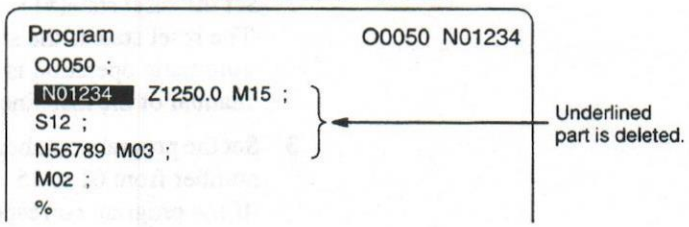
Example of deleting blocks from a block containing N01234 to a block containing N56789

Procedure

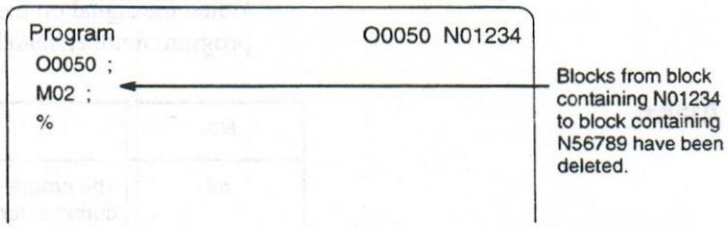
1 Search for or scan N01234.



2 Key in [N] [5] [6] [7] [8] [9] .



3 Press the [DELETE] key.



9.4 PROGRAM NUMBER SEARCH

When memory holds multiple programs, a program can be searched for. There are three methods as follows.

Procedure for program number search

Method 1

1 Select **MEMORY** mode.

2 Press [PROG] to display the program screen.

3 Key in address [O] .

4 Key in a program number to be searched for.

5 Press the **[O SRH]** key.

6 Upon completion of search operation, the program number searched for is displayed in the upper–right corner of the CRT screen
If the program is not found , P/S alarm No. 71 occurs.

Method 2

1 Select **MEMORY** mode.

2 Press [PROG] to display the program screen.

3 Press the **[O SRH]** key.
In this case, the next program in the directory is searched for .

Method 3

This method searches for the program number (0001 to 0015) corresponding to a signal on the machine tool side to start automatic operation. Refer to the relevant manual prepared by the machine tool builder for detailed information on operation.

1 Select **MEMORY** mode.

2 Set the reset state(*1)
·The reset state is the state where the LED for indicating that automatic operation is in progress is off. (Refer to the relevant manual of the machine tool builder.)

3 Set the program number selection signal on the machine tool side to a number from 01 to 15.
·If the program corresponding to a signal on the machine tool side is not registered, P/S alarm (No. 059) is raised.

4 Press the cycle start button.
·When the signal on the machine tool side represents 00, program number search operation is not performed.

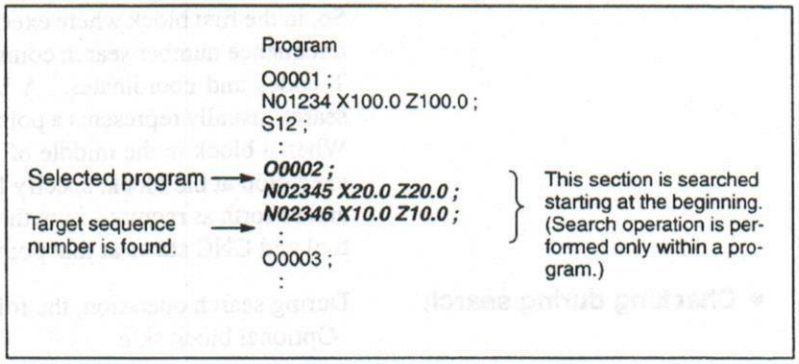
Alarm

No.	Contents
59	The program with the selected number cannot be searched during external program number search.
71	The specified program number was not found during program number search.

9.5 SEQUENCE NUMBER SEARCH

Sequence number search operation is usually used to search for a sequence number in the middle of a program so that execution can be started or restarted at the block of the sequence number.

Example) Sequence number 02346 in a program (O0002) is searched for.



Procedure for sequence number search

1 Select **MEMORY** mode.

2 Press [PROG].

3 ·If the program contains a sequence number to be searched for, perform the operations 4 to 7 below.
·If the program does not contain a sequence number to be searched for,select the program number of the program that contains the sequence number to be searched for.

4 Key in address [N].

5 Key in a sequence number to be searched for.

6 Press the **[N SRH]** key.

7 Upon completion of search operation, the sequence number searched for is displayed in the upper–right corner of the CRT screen.
If the specified sequence number is not found in the program currently selected, P/S alarm (No.060) occurs.

Explanations

• Operation during Search

Those blocks that are skipped do not affect the CNC. This means that the data in the skipped blocks such as coordinates and M, S, and T codes does not alter the CNC coordinates and modal values.
So, in the first block where execution is to be started or restarted by using a sequence number search command, be sure to enter required M, S, and T codes and coordinates. A block searched for by sequence number search usually represents a point of shifting from one process to another. When a block in the middle of a process must be searched for to restart execution at the block, specify M, S, and T codes, G codes, coordinates, and so forth as required from the MDI after closely checking the machine tool and CNC states at that point.

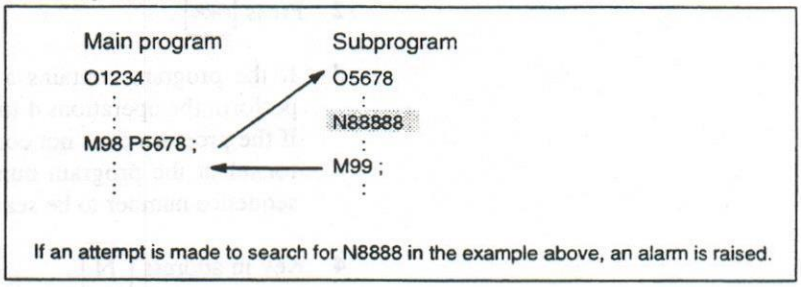
• Checking during search

During search operation, the following checks are made:
·Optional block skip

Restrictions

• Searching in sub–program

During sequence number search operation, M98Pxxxx (subprogram call) is not executed. So an alarm (No.060) is raised if an attempt is made to search for a sequence number in a subprogram called by the program currently selected.



If an attempt is made to search for N8888 in the example above, an alarm is raised.

Alarm

Number	Contents
60	Command sequence number was not found in the sequence number search.

9.6 DELETING PROGRAMS

Programs registered in memory can be deleted,either one program by one program or all at once. Also, More than one program can be deleted by specifying a range.

9.6.1 Deleting One Program

A program registered in memory can be deleted.

Procedure for deleting one program

1 Select the **EDIT** mode.

2 Press [PROG] to display the program screen.

3 Key in address [O] .

4 Key in a desired program number.

5 Press the [DELETE] key.
The program with the entered program number is deleted.

9.6.2 Deleting All Programs

All programs registered in memory can be deleted.

Procedure for deleting all programs

1 Select the **EDIT** mode.

2 Press [PROG] to display the program screen.

3 Key in address [O] .

4 Key in –9999.

5 Press edit key [DELETE] to delete all programs.

9.6.3 Deleting More Than One Program by Specifying a Range

Programs within a specified range in memory are deleted.

Procedure for deleting more than one program by specifying a range

1 Select the **EDIT** mode.

2 Press [PROG] to display the program screen.

3 Enter the range of program numbers to be deleted with address and numeric keys in the following format:
OXXXX,OYYYY
where XXXX is the starting number of the programs to be deleted and YYYY is the ending number of the programs to be deleted.

4 Press edit key [DELETE] to delete programs No. XXXX to No. YYYY.

9.7 EDITING OF CUSTOM MACROS

Unlike ordinary programs, custom macro programs are modified, inserted, or deleted based on editing units.
Custom macro words can be entered in abbreviated form.
Comments can be entered in a program.
Refer to the section 10.1 for the comments of a program.

Explanations

- **Editing unit**

When editing a custom macro already entered, the user can move the cursor to each editing unit that starts with any of the following characters and symbols:

(a) Address
(b) # located at the start of the left side of a substitution statement
(c) /, (,=, and ;
(d) First character of IF, WHILE, GOTO, END, DO, POPEN, BPRNT, DPRNT and PCLOS

On the CRT screen, a blank is placed before each of the above characters and symbols.
(Example) Head positions where the cursor is placed

```
N001 X–#100 ;
#1 =123 ;
N002 /2 X[12/#3] ;
N003 X–SQRT[#3/3*[#4+1]] ;
N004 X–#2 Z#1 ;
N005 #5 =1+2–#10 ;
IF[#1NE0] GOTO10 ;
WHILE[#2LE5] DO1 ;
#[200+#2] =#2*10 ;
#2 =#2+1 ;
END1 ;
```

- **Abbreviations of custom macro word**

When a custom macro word is altered or inserted, the first two characters or more can replace the entire word.
Namely,

WHILE → WH GOTO → GO XOR → XO AND → AN
SIN → SI COS → COTAN → TA ATAN → AT SQRT → SQ
ABS → AB BCD → BC BIN → BI FIX → FIFUP → FU
ROUND → RO END → ENPOPEN → PO BPRNT → BP
DPRNT → DP PCLOS→PC

(Example) Keying in

```
WH [AB [#2 ] LE RO [#3 ] ]
```

has the same effect as

```
WHILE [ABS [#2 ] LE ROUND [#3 ] ]
```

The program is also displayed in this way.

9.8 BACKGROUND EDITING

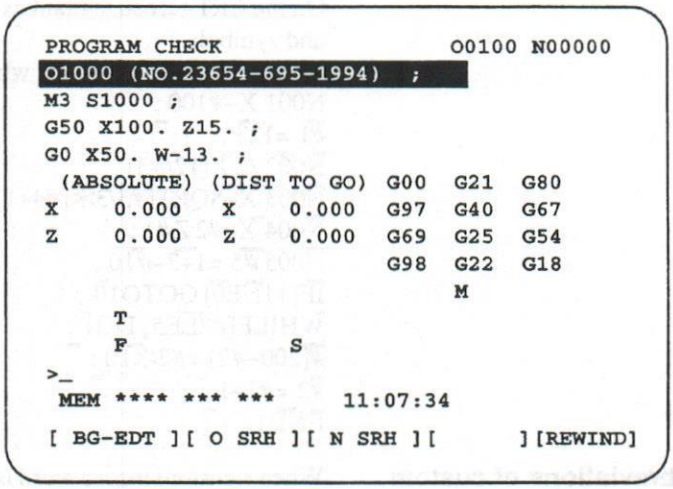
Editing a program while executing another program is called background editing. The method of editing is the same as for ordinary editing (foreground editing).
A program edited in the background should be registered in foreground program memory by performing the following operation:
During background editing, all programs cannot be deleted at once.

Procedure for background editing

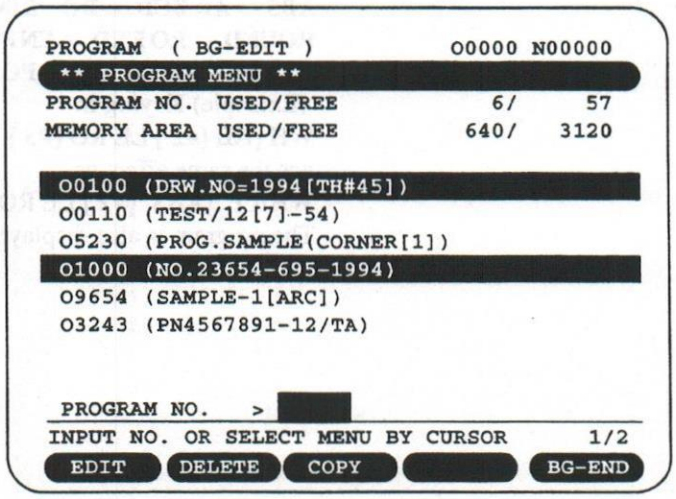
1 Enter **MEMORY** mode.
Memory mode is allowed even while the program is being executed.

2 Press function key [PROG].

3 Press the **[(OPRT)]** soft key to display the **[BG–EDT]** soft key.



4 Press the **[BG–EDT]** soft key.
The program list screen for background editing appears ("PROGRAM (BG–EDIT)" is displayed at the top left).



5 Edit a program using the background editing screen, in the same way as for ordinary editing.
The program editing procedure is the same in both the background and foreground. The number of the program being executed is dimmed on the screen and the cursor cannot be positioned to that number.

6 Once you have completed editing, press the **[BG–END]** soft key. The edited program is registered into the program memory for the foreground.

Explanation

- **Alarms during background editing**

Alarms that may occur during background editing do not affect foreground operation. Conversely, alarms that may occur during foreground operation do not affect background editing. In background editing, if an attempt is made to edit a program selected for foreground operation, a BP/S alarm (No. 140) is raised. On the other hand, if an attempt is made to select a program subjected to background editing during foreground operation (by means of subprogram calling or program number search operation using an external signal), a P/S alarm (Nos. 059, 078) is raised in foreground operation. As with foreground program editing, P/S alarms occur in background editing. However, to distinguish these alarms from foreground alarms, BP/S is displayed in the data input line on the background editing screen.

9.9 PASSWORD FUNCTION

The password function (bit 4 (NE9) of parameter No. 3202) can be locked using parameter No. 3210 (PASSWD) and parameter No. 3211 (KEYWD) to protect program Nos. 9000 to 9999. In the locked state, parameter NE9 cannot be set to 0. In this state, program Nos. 9000 to 9999 cannot be modified unless the correct keyword is set.
A locked state means that the value set in the parameter PASSWD differs from the value set in the parameter KEYWD. The values set in these parameters are not displayed. The locked state is released when the value already set in the parameter PASSWD is also set in parameter KEYWD. When 0 is displayed in parameter PASSWD, parameter PASSWD is not set.

Procedure for locking and unlocking

Locking

1 Set the MDI mode.
2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.
3 Set parameter No. 3210 (PASSWD). At this time, the locked state is set.
4 Disable parameter writing.
5 Press the RESET key to release the alarm state.

Unlocking

1 Set the MDI mode.
2 Enable parameter writing. At this time, P/S alarm No. 100 is issued on the CNC.
3 In parameter No. 3211 (KEYWD), set the same value as set in parameter No. 3210 (PASSWD) for locking. At this time, the locked state is released.
4 Set bit 4 (NE9) of parameter No. 3202 to 0.
5 Disable parameter writing.
6 Press the RESET key to release the alarm state.
7 Subprograms from program Nos. 9000 to 9999 can now be edited.

Explanations

- **Setting parameter PASSWD**

The locked state is set when a value is set in the parameter PASSWD. However, note that parameter PASSWD can be set only when the locked state is not set (when PASSWD = 0, or PASSWD = KEYWD). If an attempt is made to set parameter PASSWD in other cases, a warning is given to indicate that writing is disabled. When the locked state is set (when PASSWD = 0 and PASSWD = KEYWD), parameter NE9 is automatically set to 1. If an attempt is made to set NE9 to 0, a warning is given to indicate that writing is disabled.

- **Changing parameter PASSWD**

Parameter PASSWD can be changed when the locked state is released (when PASSWD = 0, or PASSWD = KEYWD). After step 3 in the procedure for unlocking, a new value can be set in the parameter PASSWD. From that time on, this new value must be set in parameter KEYWD to release the locked state.

- **Setting 0 in parameter PASSWD**

When 0 is set in the parameter PASSWD, the number 0 is displayed, and the password function is disabled. In other words, the password function can be disabled by either not setting parameter PASSWD at all, or by setting 0 in parameter PASSWD after step 3 of the procedure for unlocking. To ensure that the locked state is not entered, care must be taken not to set a value other than 0 in parameter PASSWD.

- **Re–locking**

After the locked state has been released, it can be set again by setting a different value in parameter PASSWD, or by turning the power to the CNC off then on again to reset parameter KEYWD.

Notes

Once the locked state is set, parameter NE9 cannot be set to 0 and parameter PASSWD cannot be changed until the locked state is released or the memory all–clear operation is performed. Special care must be taken in setting parameter PASSWD.

10 CREATING PROGRAMS

You can create a program by applying any of the following methods:

- Creating a program by guidance programming
- Creating a CNC language program with the MDI panel
- Creating a program in **TEACH IN** mode
- Creating a program with the FANUC SYSTEM P automatic programming system

This chapter describes creating a program by guidance programming, creating a CNC language program with the MDI panel, and creating a program in **TEACH IN** mode. This chapter also describes the automatic insertion of sequence numbers.

10.1 CREATING A PROGRAM WITH GUIDANCE PROGRAMMING

This section describes how to create and register a program by means of the optional guidance programming function.

10.1.1 Program Menu Screen

The program menu screen is displayed by following the procedure below

Procedure

1 Select **EDIT** mode.

2 Press the PROG function key or the **[PRGRM]** soft key. The program menu screen is displayed.

```
PROGRAM  ( EDIT )                    O9654 N00000
** PROGRAM MENU **
PROGRAM NO.  USED/FREE                 5/     58
MEMORY AREA  USED/FREE               400/   3280

O1002 (NO. 4568-256-47893)
 O0100 (#23546-21)
 O0110 (TEST/12[7]-54)
 O9654 (SAMPLE-1[ARC])
 O3243 (PN4567891-12/TA)


 PROGRAM NO.   >
INPUT NO. OR SELECT MENU BY CURSOR       1/2
 EDIT  DELETE  COPY
```

```
PROGRAM  ( EDIT )                    O9654 N00000
** PROGRAM MENU **
PROGRAM NO.  USED/FREE                 5/     58
MEMORY AREA  USED/FREE               480/   3280

O1002 (NO. 4568-256-47893)
 O0100 (#23546-21)
 O0110 (TEST/12[7]-54)
 O9654 (SAMPLE-1[ARC])
 O3243 (PN4567891-12/TA)


 PROGRAM NO.   >
INPUT NO. OR SELECT MENU BY CURSOR       2/2
 PUNCH  READ                          COMENT
```

The program menu screen displays seven lines. These lines can be scrolled up and down by pressing the ↑ or ↓ key.

From the program menu screen, the following operations can be selected by pressing the corresponding soft keys. If the rightmost soft key is pressed, the subsequent page is displayed.

Soft key	Page	Function
EDIT	1/2	Registers and edits a program.
DELETE		Deletes a program.
COPY		Copies a program.
PUNCH	2/2	Outputs a program, through the reader/punch interface, to an external memory unit.
READ		Reads a program, through the reader/punch interface, from an external memory unit and registers that program in a part program storage area.
COMMENT		Selects whether to display a program name.

Description

- **For an explanation of the use of the [DELETE] and [COPY] soft keys** — See Chapter 9.
- **For an explanation of the use of the [PUNCH] and [READ] soft keys** — See Section 8.4.

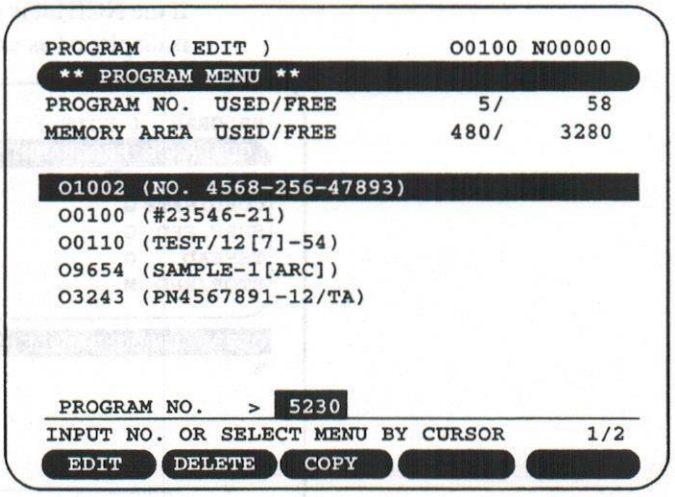
10.1.2 Registering a Program

When the program menu screen is displayed, a guidance message prompting the operator to input a program number is displayed.

A program can be registered by following the procedure below

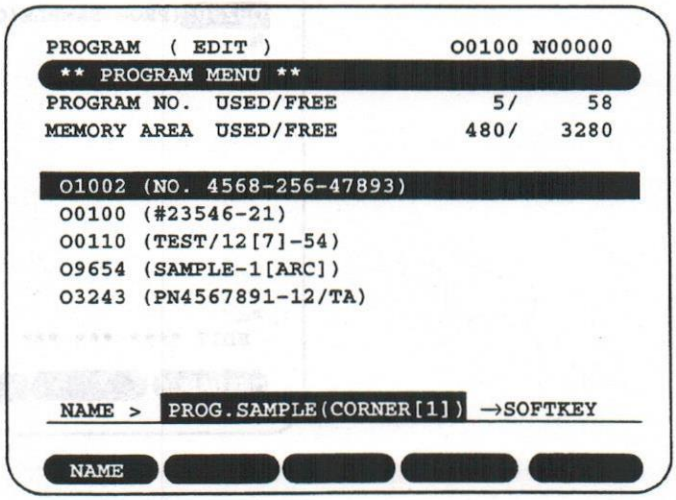
Procedure

1 Using the numeric keypad, enter a program number of up to four digits.
Specify a number which is not yet registered.



2 Press the **[EDIT]** soft key.
Check that the data protect key on the operator's panel is released.

3 The **[NAME]** soft key is displayed and the guidance message for entering a program name is displayed.
Using the address keys and numeric keypad, enter a desired program name of up to 23 characters. Then, press the **[NAME]** soft key.
Do not include the EOB (;) in the program name.



If a name does not have to be registered, press the **[NAME]** soft key directly.

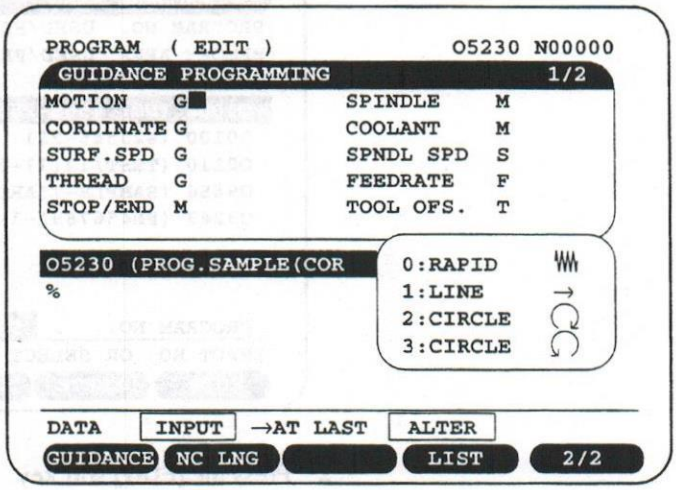
4 The guidance programming screen or CNC language screen is displayed. The screen to be displayed depends on the setting of the following parameter:

Bit 0 of Parameter 9320 (NCE)

0: Displays the guidance programming screen after a program number is registered.

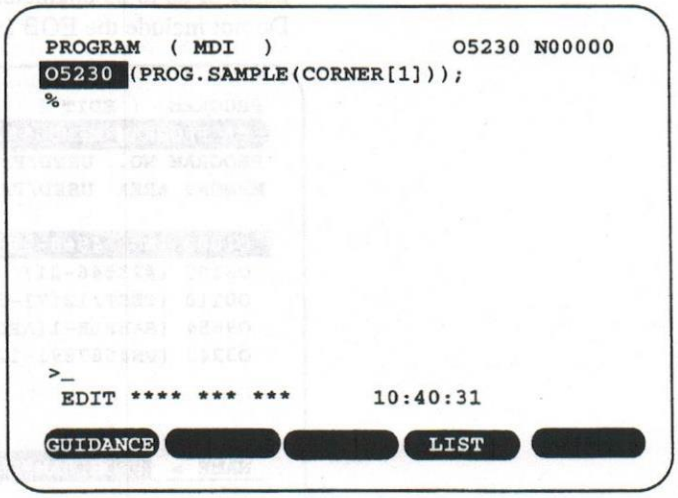
1: Displays the CNC language screen after a program number is registered.

If the NCE bit is set to 0, the guidance programming screen is displayed as shown below:



To display the CNC language screen, press the **[NC LNG]** soft key.

If the NCE bit is set to 1, the CNC language screen is displayed as shown below:



To display the guidance programming screen, press the **[GUIDANCE]** soft key.

To return to the program menu screen, press the **[LIST]** soft key.

5 Creating a program.

For details of creating a program by means of guidance programming, see Section 10.1.4.

For details of creating a program in the CNC language, see Section 9.1.8.

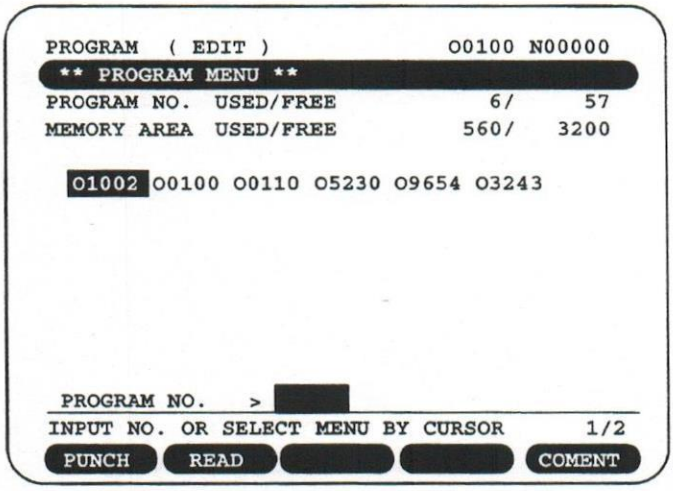
10.1.3 Comment

The operator can select whether to display program names on the program menu screen.

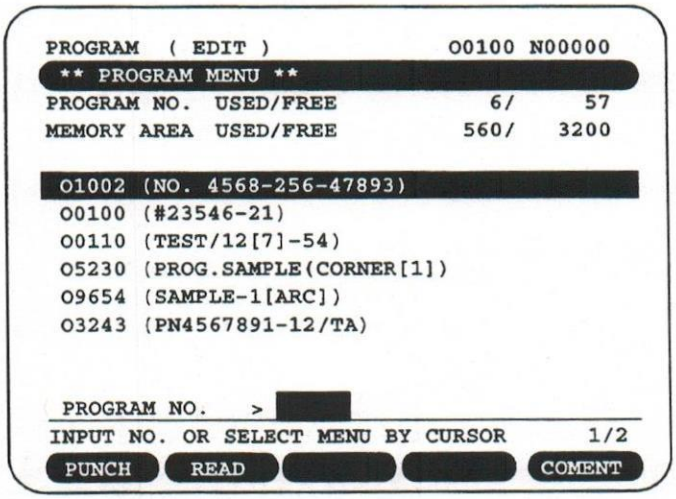
The comment display can be switched by following the procedure below

Procedure

1 Press the rightmost soft key. The **[COMMENT]** soft key is displayed. If the **[COMMENT]** soft key is pressed while the program names are displayed on the program menu screen, the program names are hidden and only the program numbers are displayed.


PROGRAM (EDIT) O0100 N00000
** PROGRAM MENU **
PROGRAM NO. USED/FREE 6/ 57
MEMORY AREA USED/FREE 560/ 3200
O1002 O0100 O0110 O5230 O9654 O3243
PROGRAM NO. >
INPUT NO. OR SELECT MENU BY CURSOR 1/2
PUNCH READ COMENT

2 If the **[COMMENT]** soft key is pressed while only the program numbers are displayed on the program menu screen, the program names are displayed on the screen.



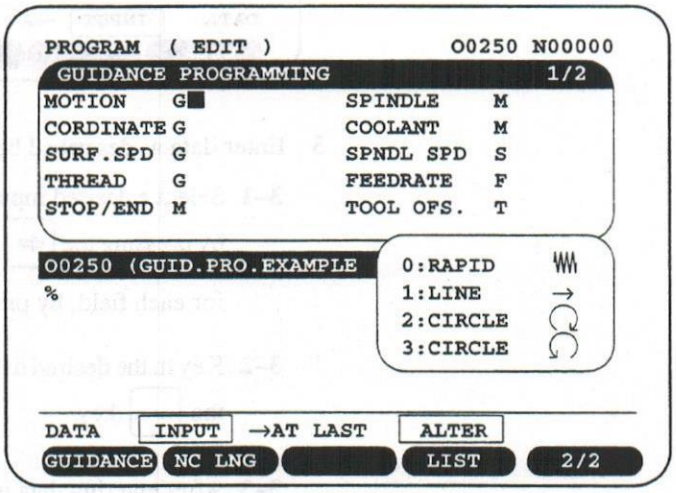
10.1.4 Guidance Programming

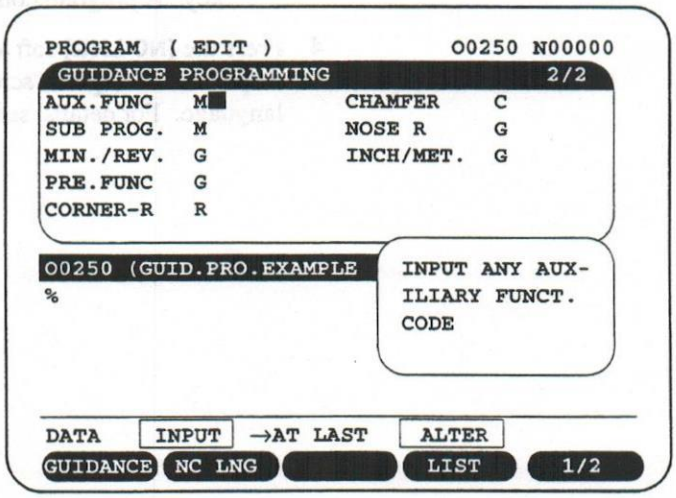
The guidance programming function automatically creates a program in the CNC language using data entered by the operator in response to prompts displayed on the screen. Thus, even a user with little experience of CNC can easily create a program.

Creating a program by means of guidance programming

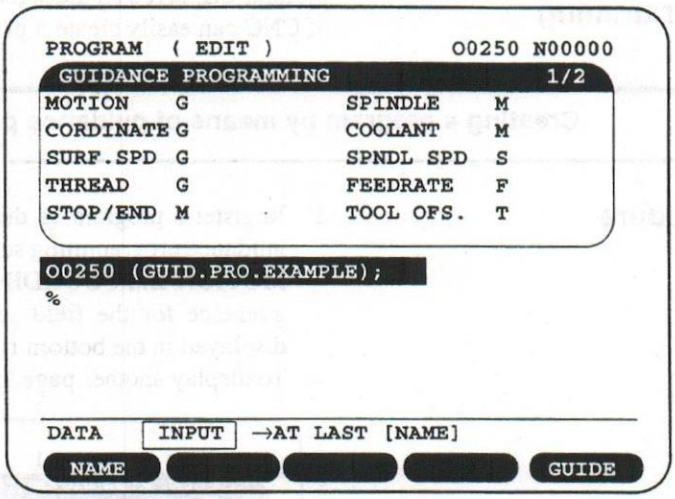
Procedure

1 Register a program as described in Section 10.1.2. The following guidance programming screen is displayed. The input fields, such as MOTION and COORDINATE, are displayed over two pages. The guidance for the field selected by positioning the cursor is also displayed in the bottom right frame.
To display another page, press the **[1/2]** or **[2/2]** soft key.





2 If the rightmost soft key is pressed, the secondary soft keys are displayed. To clear the guidance from the screen, press the **[GUIDE]** soft key. To display the guidance, press the same soft key again.



3 Enter data as described below:

3–1 Select a desired input field such as MOTION or CORDINATE by pressing the ← or → key. Select the data entry position for each field, by pressing the ↑ or ↓ key.

3–2 Key in the desired data as explained in the guidance. Then, press the INPUT key.

3–3 After entering data into the desired input fields, press the INSERT key. A program consisting of a single block is created.

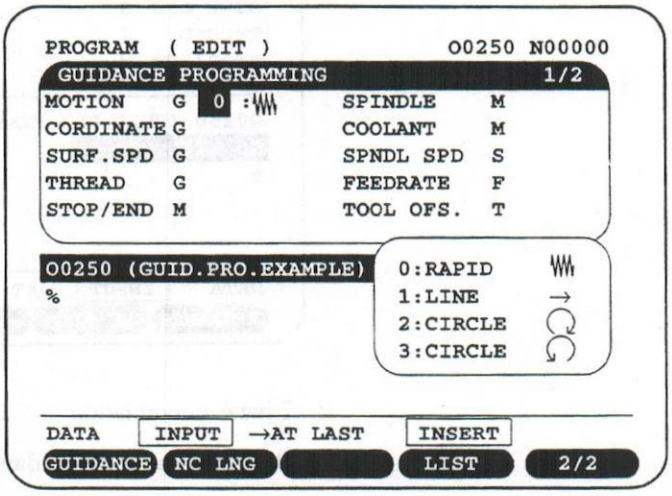
4 Press the **[NC LNG]** soft key. The CNC language program screen is displayed. Using this screen, a program can be created in the CNC language. For details, see Chapter 9.

Programming a move command

Procedure

1 Positioning

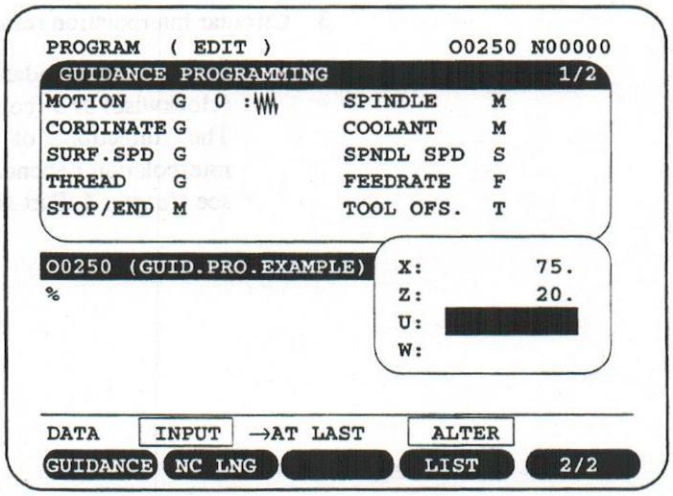
1–1 Following the guidance displayed for MOVE, enter a value of 0.



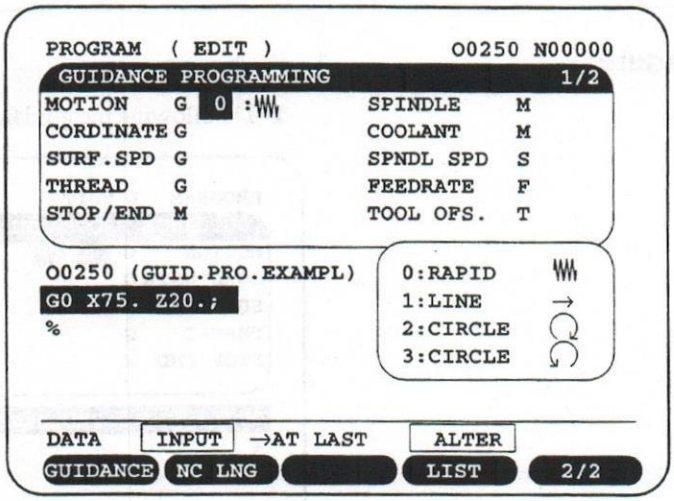
1–2 The guidance screen for entering an end point is displayed. Enter the following:

Coordinates of the end point :
X, Z (in the absolute mode)

Amount of travel from the current position :
U, W (in incremental mode)



1–3 Press the INSERT key. A positioning program is created.



2 Linear interpolation

2–1 Following the guidance displayed for MOVE, enter a value of 1.

2–2 The guidance screen for entering an end point is displayed. Enter the following:

Coordinates of the end point :
X, Z (in absolute mode)

Amount of travel from the current position :
U, W (in incremental mode)

2–3 Enter data into the MOVE field.

3 Circular interpolation (clockwise, counterclockwise)

3–1 Following the guidance displayed for MOVE, enter a value of 2 (clockwise) or 3 (counterclockwise).
The direction of the clockwise or counterclockwise interpolation depends on the coordinate system. (For details, see Chapter 4, Part II.)

3–2 The guidance screen for entering the end and center of an arc is displayed. Enter the following:

In absolute mode

Input item	Description
X	X coordinate of the end point
Z	Z coordinate of the end point
R	Radius of the arc
I	Distance from the start point to the center in the X direction (signed)
K	Distance from the start point to the center in the Z direction (signed)

In incremental mode

Input item	Description
U	Distance from the start point to the end point in the X direction
W	Distance from the start point to the end point in the Z direction
R	Radius of the arc
I	Distance from the start point to the center in the X direction (signed)
K	Distance from the start point to the center in the Z direction (signed)

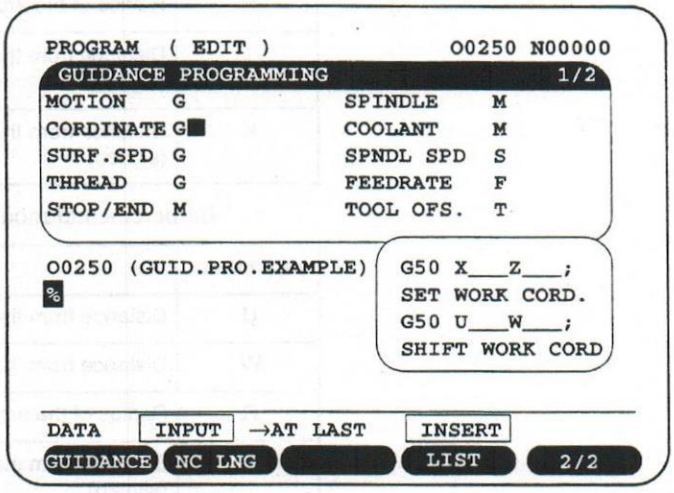
3–3 Enter the data in the MOVE field.

Specifying the coordinate system

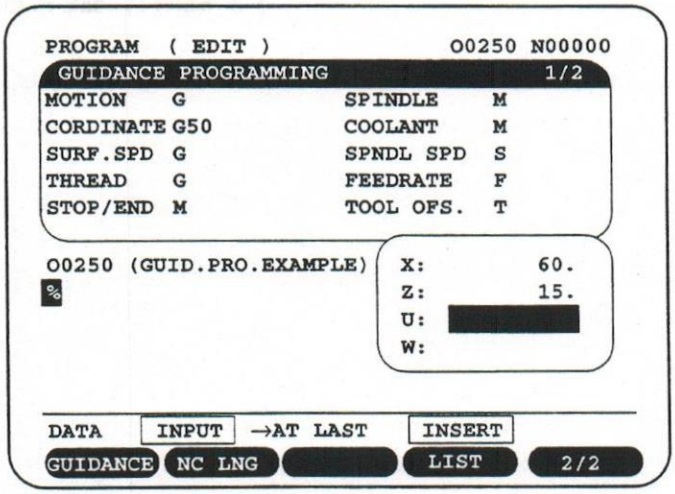
Specify the workpiece coordinate system to be used for machining or a shift of the coordinate system, as described below. (For details, see Section 7.7.2, Part II.)

Procedure

1 Enter value of 50 in the COORDINATE field.



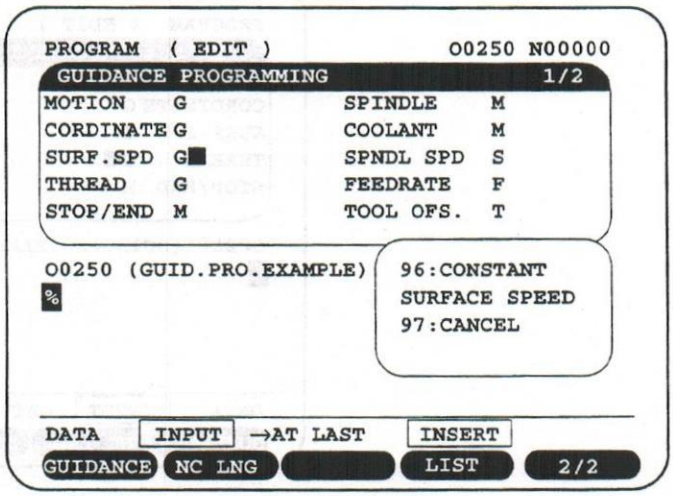
2 Enter the set value for the workpiece coordinate system (X, Z) or the shift of the coordinate system (U, W).



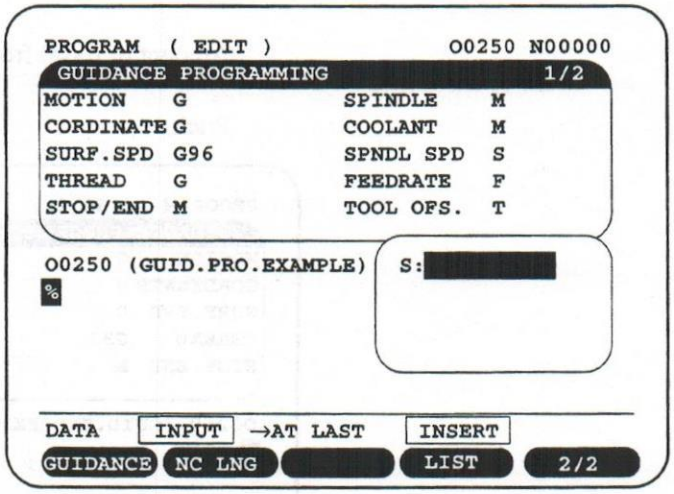
Specifying constant surface speed control

Procedure

1 On the guidance screen, enter a value of 96 (constant surface speed control) or 97 (constant surface speed cancellation) in the SURF.SPD field.



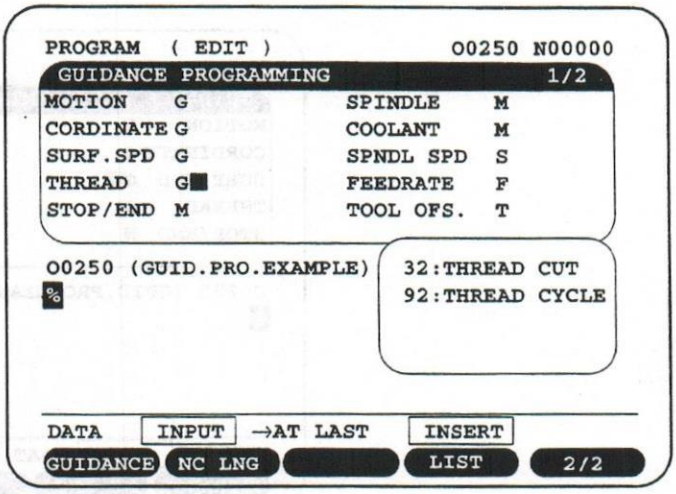
2 If 96 is entered, the screen for entering the surface speed is displayed. Enter a desired surface speed.



Threading programming

Procedure

1 On the guidance screen, enter a value of 32 (threading) or 92 (threading cycle) in the THREAD field.

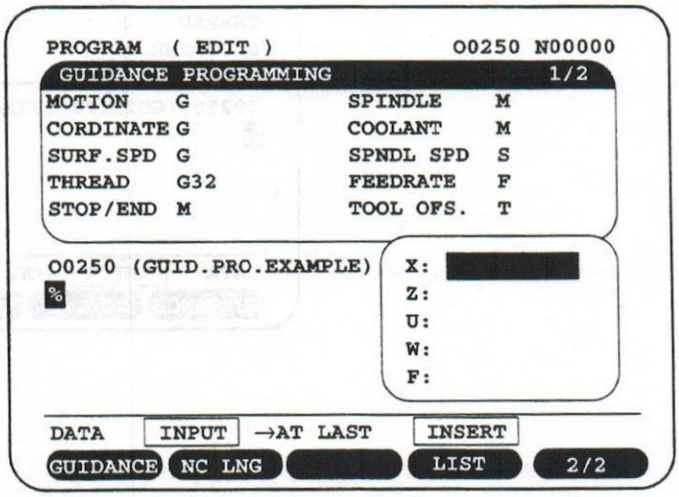


2 If a value of 32 is entered, the guidance screen for entering the end point and pitch is displayed. Enter the following:

Coordinates of the end point :
X, Z (in absolute mode)

Amount of travel from the current position :
U, W (in incremental mode)

Pitch : F



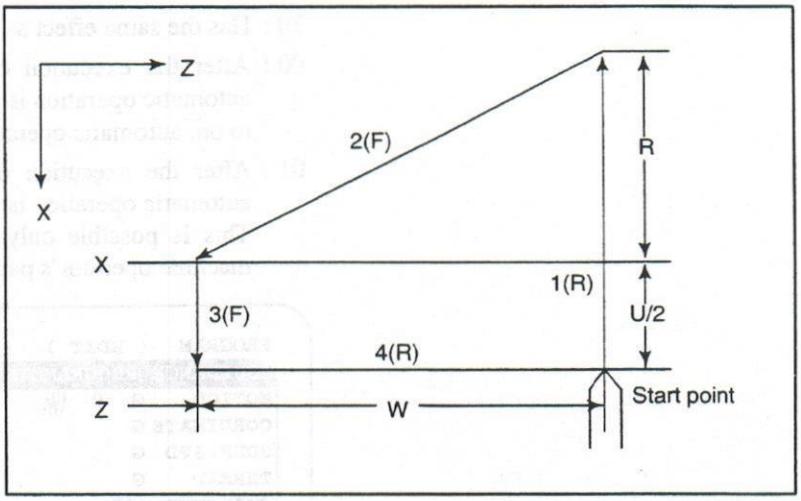
3 If a value of 92 is entered, enter the following data for the threading cycle:

Coordinates of the thread end point :
X, Z (in absolute mode)

Amount of travel from the current position to the thread end point :
U, W (in incremental mode)

Pitch : F

Radial difference between the start point and end point :
R (for tapered threading)



(For details, see Sections 4.4 and 13.1, Part II.)

Specifying an end/stop

Specify the command to end or stop a program as described below:

Procedure

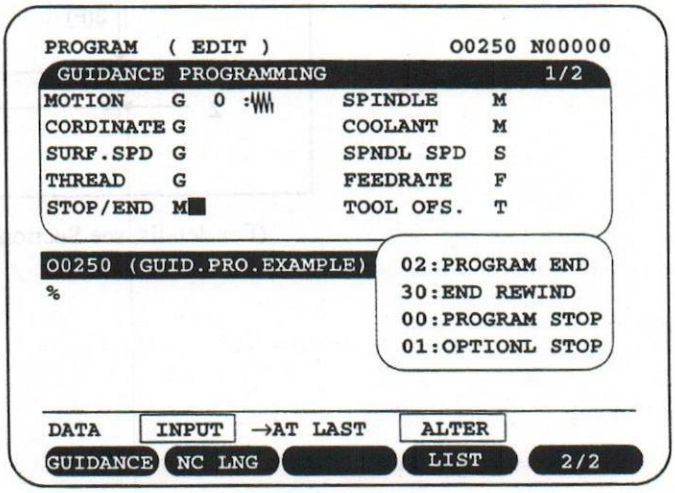
1 In the STOP/END field, enter one of the following values. Each value represents a function to end or stop a program.

02 : Represents the end of the main program.
Automatic operation is stopped and reset. (This operation may vary with the machine tool builder.)
Control is returned to the beginning of the program.

30 : Has the same effect as 02 above.

00 : After the execution of a block in which M00 is specified, automatic operation is stopped. When the start signal is next set to on, automatic operation is resumed.

01 : After the execution of a block in which M01 is specified, automatic operation is stopped.
This is possible only when the optional stop switch on the machine operator's panel is set to on.

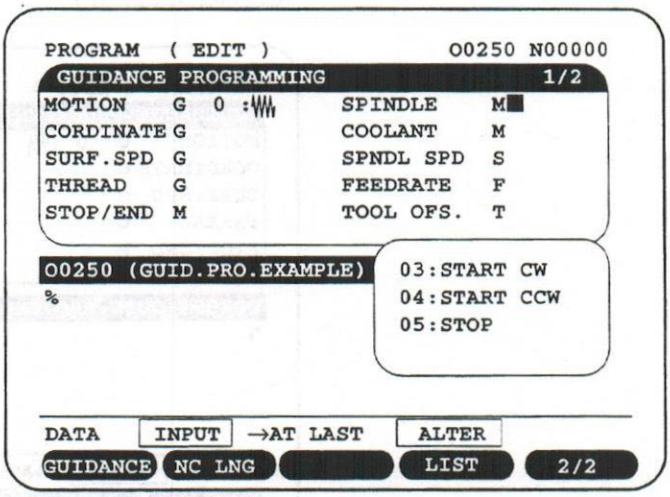


Specifying the spindle command

Specify the command to be used to start or stop the spindle as described below:

Procedure

1 On the guidance screen, enter a value of 03 (forward rotation), 04 (backward rotation), or 05 (stop) in the SPINDLE field.

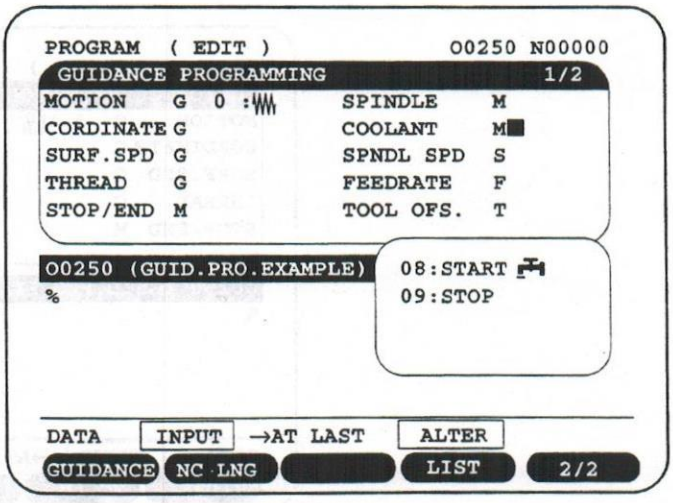


Coolant programming

Specify the command to open or close the coolant valve.

Procedure

1 On the guidance screen, enter value of 08 (start) or 09 (stop) in the COOLANT field.

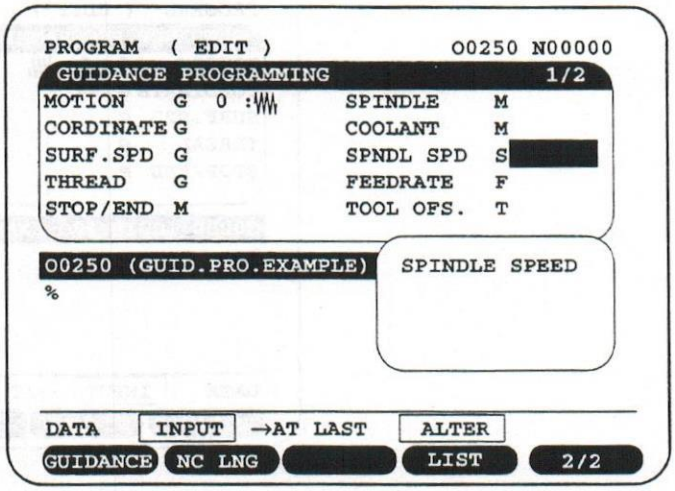


Spindle speed programming

Specify the spindle speed command or circumference speed command for constant circumference speed control.

Procedure

1 Enter a value for the rotational speed (rpm) or circumference speed (m/min or feet/min) in the SPNDL SPD field. (For details, see Chapter 9, Part II.)

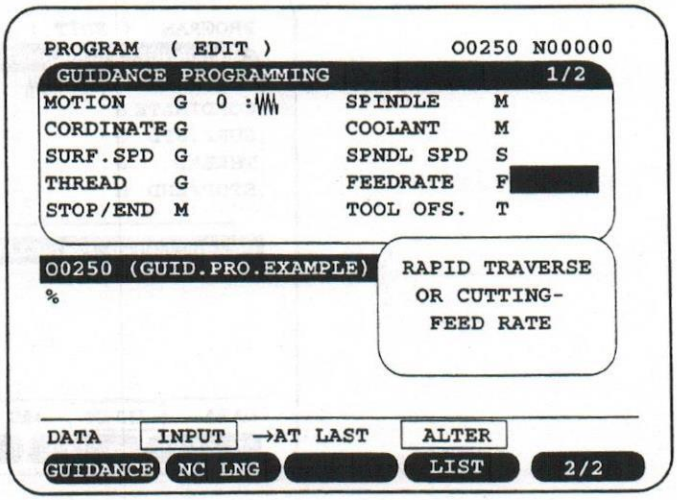


Feedrate programming

Specify the cutting feedrate command.

Procedure

1 Enter a value for the feedrate in the FEEDRATE field. (For details, see Chapter 5, Part II.)



Tool offset programming

Procedure

Specify the tool offset number command.

Tool commands are specified using two or four digits. The type of command to be used varies with the machine. Check the manual provided by the machine tool builder for details of the command type.

Machine using a tool command with two digits:

Specify two digits "Δ □" in the TOOL OFS. field.

Δ : Specify a tool number.

□ : Specify an offset number.

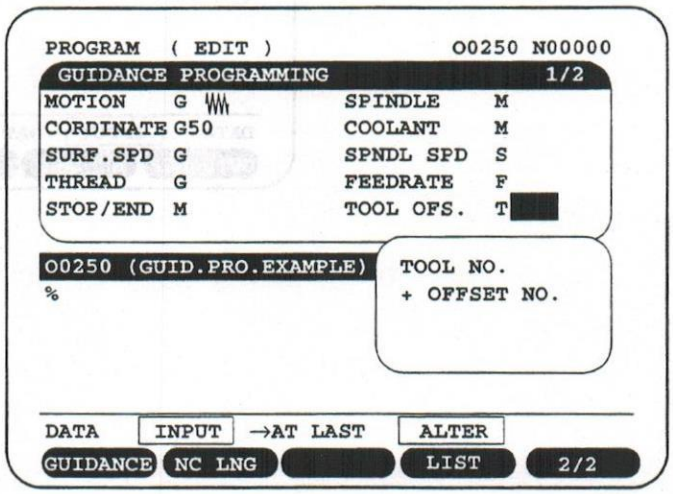
Machine using a tool command with four digits:

Specify four digits "Δ Δ □ □" in the TOOL OFS. field.

Δ Δ : Specify a tool number.

□ □ : Specify an offset number.

(For details, see Section 14.1, Part II.)

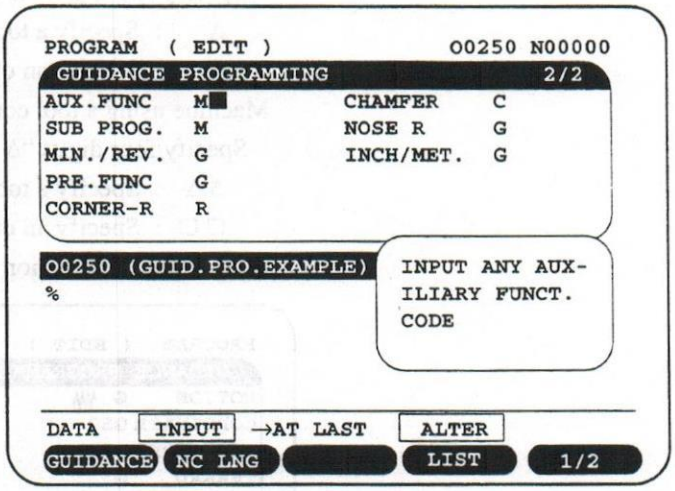


Optional M code programming

Specify a function (M code) other than a function related to end/stop, spindle command, coolant, or subprogram call.

Procedure

1 Enter an M code number directly in the AUX.FUNC field. For details of the correspondence between the M code number and function, see the manual provided by the machine tool builder.

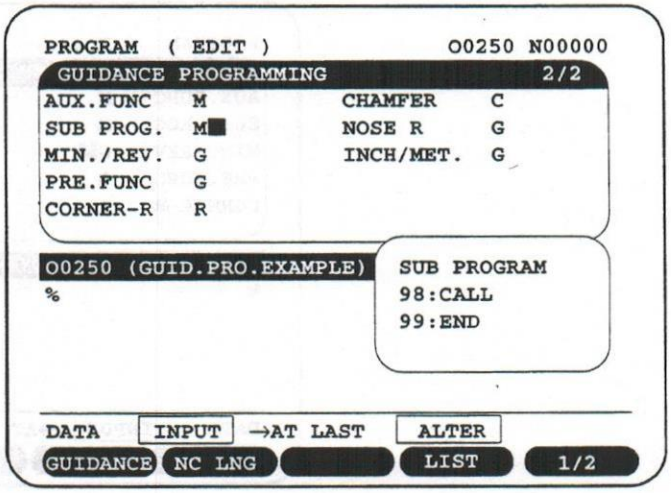


Subprogram call

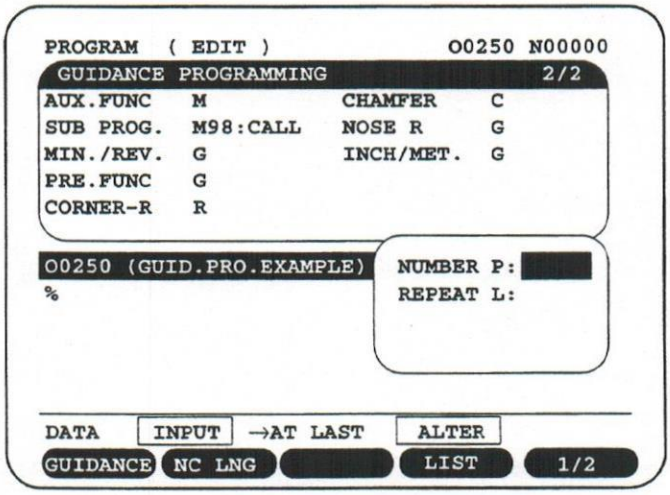
Procedure

1 Calling a subprogram
Specify the command used to call a subprogram from a main program.

1–1 Enter 98 (subprogram call) in the SUB PROG. field.



1–2 In the guidance frame, enter the desired subprogram number and the number of calls in the NUMBER P: and REPEAT L: fields, respectively.



2 Ending a subprogram

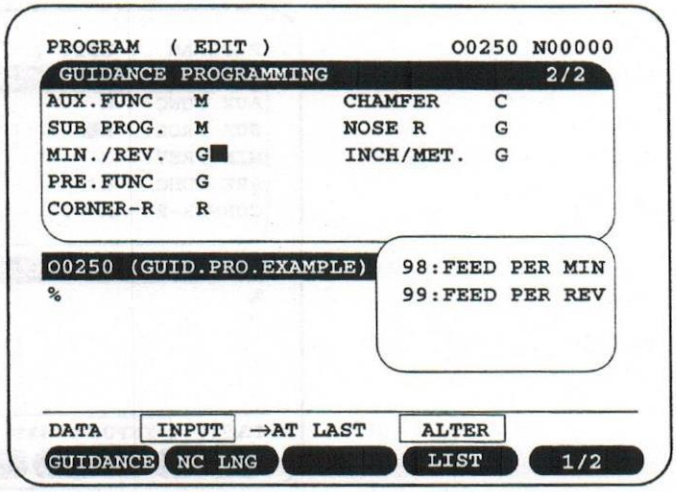
2–1 Enter 99 (end of subprogram) in the SUB PROG. field.
This command returns control from the subprogram to the main program.
(For details, see Chapter 12, Part II.)

Specifying feed mode

Specify the command used to select feed–per–minute mode or feed–per–rotation mode for cutting feed.

Procedure

1 Enter 98 (feed per minute) or 99 (feed per rotation) in the MIN./REV. field. (For details, see Section 5.3, Part II.)

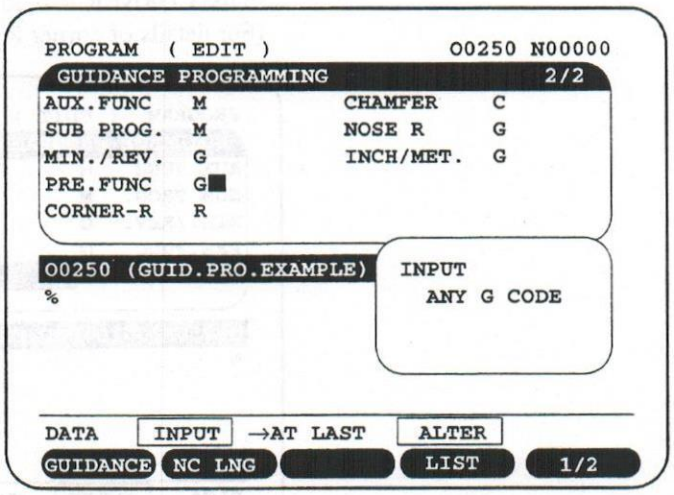


Optional G code programming

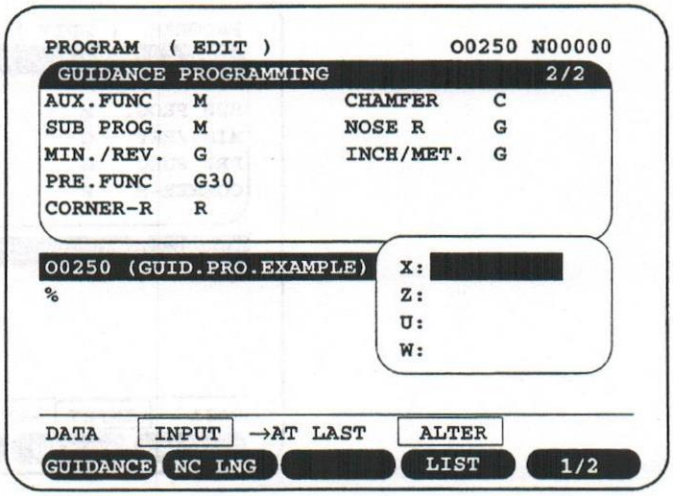
Specify a command used to program the G code, other than a code related to the move command, coordinate system, constant surface speed control, threading, feed mode, tool–tip radius compensation, or input system.

Procedure

1 Enter a desired G code number directly in the PRE.FUNC field. (For details of the correspondence between the G code number and function, see Chapter 3, Part II.)



2 Enter a G code. The guidance frame for entering coordinates is displayed. If required, enter the necessary data.

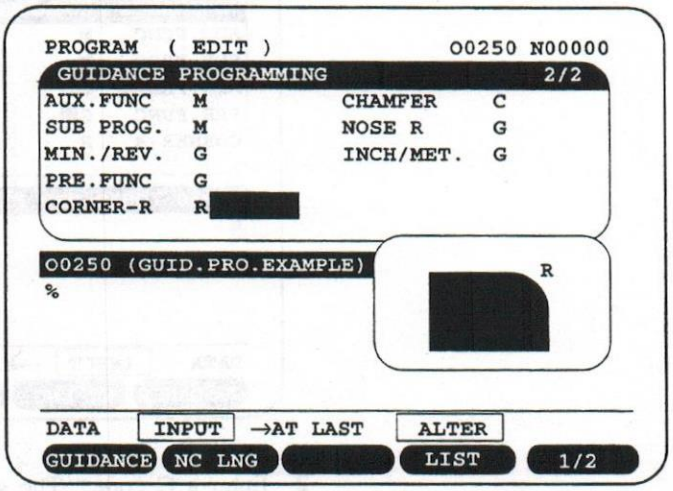


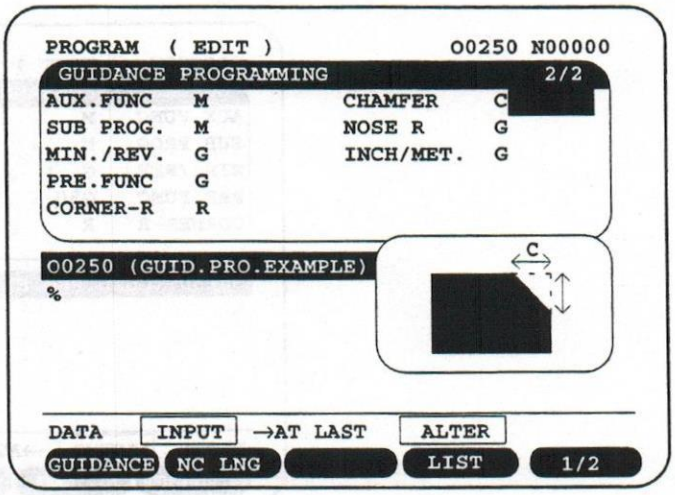
Corner R or C programming

Specify the command used to insert a corner R or C between cutting commands of linear or circular interpolation.

Procedure

1 Enter a value directly in the CORNER–R or CHAMFER field.
If the value is entered in the CORNER R field, nothing can be entered in the CHAMFER field.
If the value is entered in the CHAMFER field, nothing can be entered in the CORNER R field.
(For details of corner R and corner C, see Section 13.2, Part II.)





Specifying tool–tip radius compensation

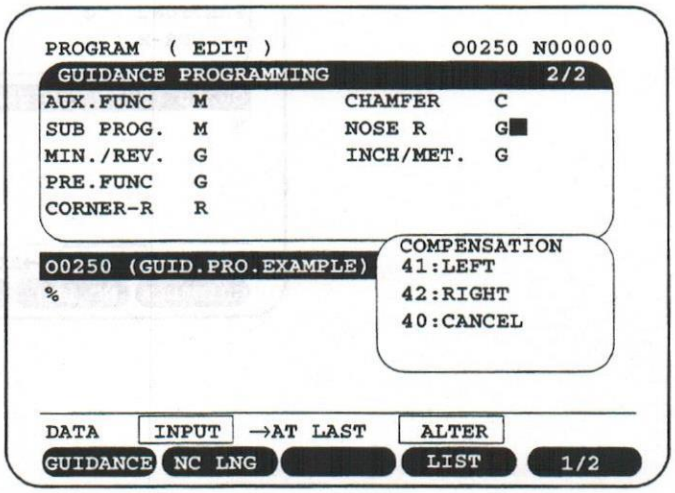
Specify the function to be used to compensate for figure errors caused by the radius of the tool tip, as described below.

Procedure

1 Specifying tool–tip radius compensation
Enter one of the following values in the NOSE R field:

41 : Moves the tool to the left of the programmed tool path.
42 : Moves the tool to the right of the programmed tool path.

The right and left sides of the programmed tool path vary with the coordinate system.
(For details, see Section 14.2, Part II.)



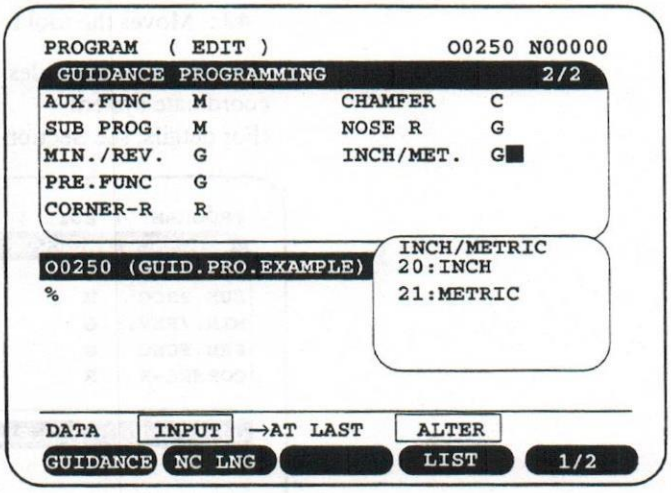
2 Canceling tool–tip radius compensation
Enter value 40 (cancel) in the NOSE R field.

Specifying input system

Specify the function to be used to select the inch input system or metric input system.

Procedure

1 Enter 20 (inch data input) or 21 (metric data input) in the INCH/MET. field.
(For details, see Section 8.2, Part II.)



10.1.5 Warning Messages in Guidance Programming

The following warning messages can be displayed in guidance programming:

Message to be displayed	Description and action to be taken
INPUT GUIDANCE NUMBER	In an input field of the guidance programming screen, a value which is not displayed in the guidance frame was entered. Select and enter one of the values provided in the guidance frame.
THE PROGRAM IS PROTECTED	The data protect key on the machine operator's panel is set to on. Or, the memory protect switch on the software operator's panel is set to the PROTECT position. To create or modify a program, release the memory protection.
CALCULATION ERROR	The pocket calculator function cannot handle the specified value. Enter a valid value.
EDITING IMPOSSIBLE	An attempt was made to modify or edit a program, but there is insufficient free space in memory. Delete any unnecessary programs, then retry the operation.

If an alarm is generated during the operation, the following message is displayed:

Message to be displayed	Description and action to be taken
AN ALARM OCCURS	A P/S or BP/S alarm occurred. Display the alarm message screen and check the details.

Description

- **Automatic switching to alarm message screen**

  Set the NPA bit (bit 7 of parameter 3111) to 0. The alarm message is automatically displayed if an alarm is generated.

10.2 CREATING A PROGRAM IN THE CNC LANGUAGE ON THE MDI PANEL

A program can be created in **EDIT** mode, using the program editing function described in Chapter 9.

A program is created in the CNC language on the MDI panel by following the procedure below

Procedure

1 Select **EDIT** mode.

2 Press the [PROG] key or **[PRGRM]** soft key. The program menu screen is displayed.

3 Using the numeric keypad, enter the desired program number.

4 Press the **[EDIT]** soft key.

5 A guidance line for entering a program name is displayed. If required, enter a program name.

6 Press the **[NAME]** soft key.

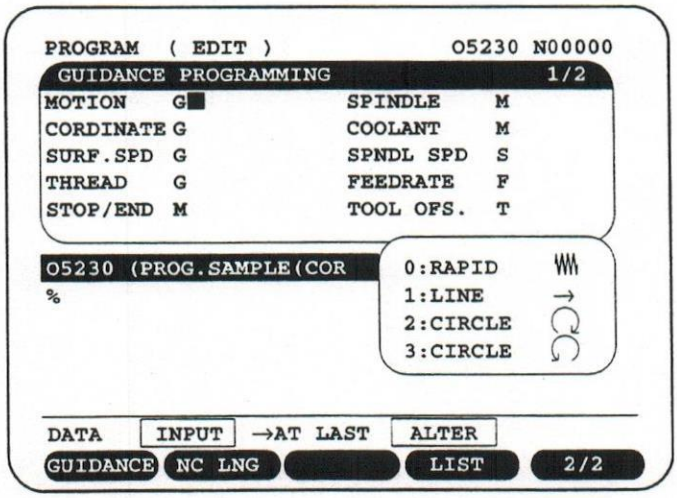
7 The guidance programming screen or CNC language screen is displayed. The screen to be displayed varies with the specification of this parameter:

Bit 0 of parameter 9320 (NCE)

0: Displays the guidance programming screen after a program number is registered.

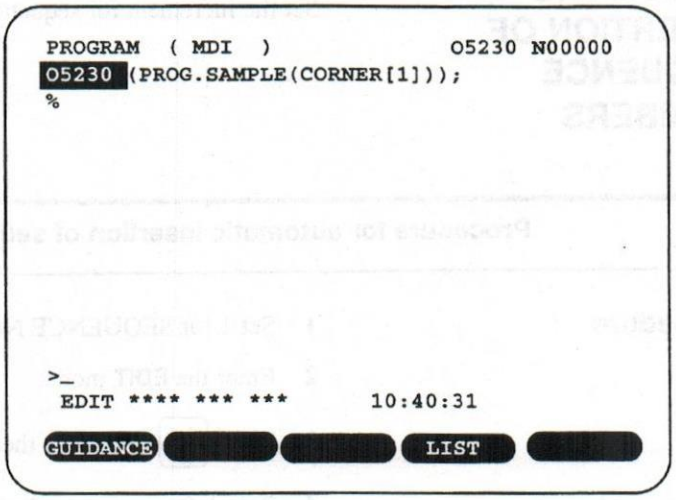
1: Displays the CNC language screen after a program number is registered.

If the NCE bit is set to 0, the guidance programming screen is displayed as shown below:



To display the CNC language screen, press the **[NC LNG]** soft key.

If the NCE bit is set to 1, the CNC language screen is displayed as shown below:



To display the guidance programming screen, press the **[GUIDANCE]** soft key.

8 On the CNC language screen, create a program as described in Chapter 9 (Sections 9.2 to 9.7).

Description

- **Program comment**

A comment describing a program can be coded with control out/in.

Example) O0001 (FANUC SERIES 20);
M08 (COOLANT ON);

- To register a comment, key in control out (, comment, and control in), then press edit key [INSERT].
- If the [INSERT] key is pressed before completing specification of the comment, and the subsequent part is registered in two or more installments, the entered data is checked as in normal editing. The comment may not have been registered as desired. (The entered comment cannot be input or can be changed or deleted.)

Note the following restrictions, imposed on the subsequent part of the comment:

- Only control in) cannot be registered.
- The subsequent part of a comment cannot be registered if it begins with a number, space or address O.
- If an abbreviation for a macro is entered, it is converted and registered in the macro language. (See Section 9.7, Part III.)
- Although address O, a number following the address, and a space can be keyed in, they are deleted when the comment is registered.

- **Detailed procedure for displaying the CNC language screen**

For details of the procedure used to display the CNC language screen, see Sections 10.1.1 and 10.1.2.

10.3 AUTOMATIC INSERTION OF SEQUENCE NUMBERS

Sequence numbers can be automatically inserted in each block when a program is created using the MDI keys in the EDIT mode.
Set the increment for sequence numbers in parameter 3216.

Procedure for automatic insertion of sequence numbers

Procedure

1 Set 1 for SEQUENCE NO. (see subjection 11.4.5).

2 Enter the **EDIT** mode.

3 Press [PROG] to display the program screen.

4 Search for or register the number of a program to be edited and move the cursor to the EOB (;) of the block after which automatic insertion of sequence numbers is started.
When a program number is registered and an EOB (;) is entered with the [INSERT] key, sequence numbers are automatically inserted starting with 0. Change the initial value, if required, according to step 10, then skip to step 7.

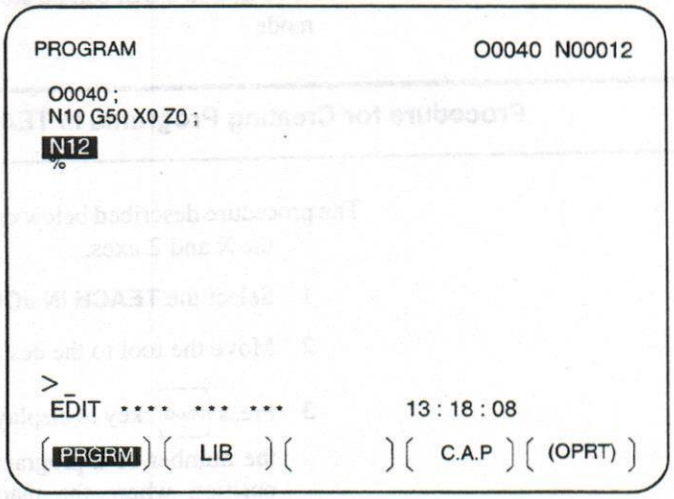
5 Set the intial value.
a. Display the CNC language screen.
b. Press address key [N] and enter the initial value of N.

6 Press [INSERT] key.

7 Enter each word of a block.

8 Press [EOB] key.

9 Press [INSERT]. The EOB is registered in memory and sequence numbers are automatically inserted. For example, if the initial value of N is 10 and the parameter for the increment is set to 2, N12 inserted and displayed below the line where a new block is specified.



10 ·In the example above, if N12 is not necessary in the next block, pressing the [DELETE] key after N12 is displayed deletes N12.

·To insert N100 in the next block instead of N12, enter N100 and press [ALTER] after N12 is displayed. N100 is registered and initial value is changed to 100.

10.4 CREATING PROGRAMS IN TEACH IN MODE

When the playback option is selected, the **TEACH IN JOG** mode and **TEACH IN HANDLE** mode are added. In these modes, a machine position along the X and Z axes obtained by manual operation is stored in memory as a program position to create a program.
The words other than X, Z, and Y, which include O, N, G, R, F, C, M, S, T, P, Q, and EOB, can be stored in memory in the same way as in **EDIT** mode.

Procedure for Creating Programs in TEACH IN Mode

The procedure described below can be used to store a machine position along the X and Z axes.

1 Select the **TEACH IN JOG** mode or **TEACH IN HANDLE** mode.

2 Move the tool to the desired position with jog or handle.

3 Press [PROG] key to display the program screen. Search for or register the number of a program to be edited and move the cursor to the position where the machine position along each axis is to be registered (inserted).

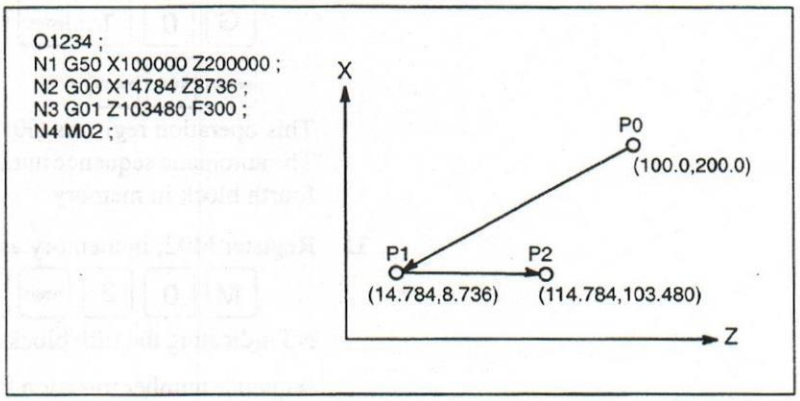
4 Key in address [X].

5 Press the [INSERT] key. Then a machine position along the X axis is stored in memory.
(Example) X10.521 Absolute position (for mm input)
X10521 Data stored in memory

6 Similarly, key in [Z], then press the [INSERT] key. Then a machine position along the Z axis is stored in memory.

All coordinates stored using this method are absolute coordinates.

Examples



1 Set the setting data SEQUENCE NO. to 1 (on). (The incremental value parameter (No. 3216) is assumed to be "1".)

2 Select the **TEACH IN HANDLE** mode.

3 Make positioning at position P0 by the manual pulse generator.

4 Select the program screen.

5 Enter program number O1234 as follows:

[O] [1] [2] [3] [4] [INSERT]

This operation registers program number O1234 in memory. Next, press the following keys:

[EOB] [INSERT]

An EOB (;) is entered after program number O1234. Because no number is specified after N, sequence numbers are automatically inserted for N0 and the first block (N1) is registered in memory.

6 Enter the P0 machine position for data of the first block as follows:

[G] [5] [0] [INSERT] [X] [INSERT] [Z] [INSERT] [EOB] [INSERT]

This operation registers G50 X100000 Z200000 ; in memory. The automatic sequence number insertion function registers N2 of the second block in memory.

7 Position the tool at P1 with the manual pulse generator.

8 Enter the P1 machine position for data of the second block as follows:

[G] [0] [0] [INSERT] [X] [INSERT] [Z] [INSERT] [EOB] [INSERT]

This operation registers G00 X14784 Z8736; in memory. The automatic sequence number insertion function registers N3 of the third block in memory.

9 Position the tool at P2 with the manual pulse generator.

10 Enter the P2 machine position for data of the third block as follows:

[G] [0] [1] [INSERT] [Z] [INSERT] [F] [3] [0] [0]
[INSERT] [EOB] [INSERT]

This operation registers G01 Z103480 F300; in memory.
The automatic sequence number insertion function registers N4 of the fourth block in memory.

11 Register M02; in memory as follows:

[M] [0] [2] [INSERT] [EOB] [INSERT]

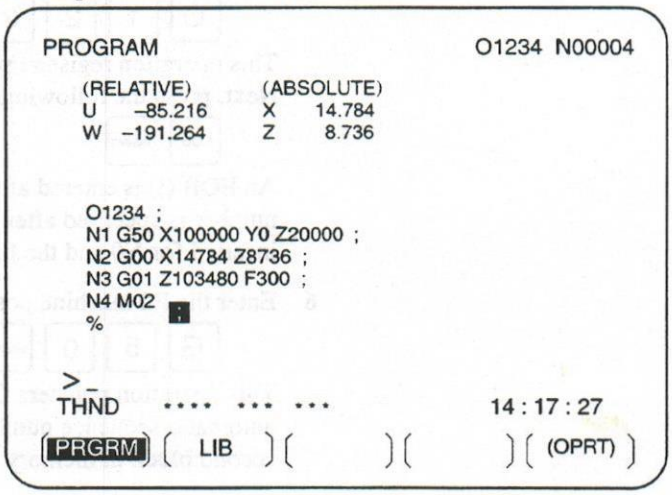
N5 indicating the fifth block is stored in memory using the automatic sequence number insertion function. Press the [DELETE] key to delete it.

This completes the registration of the sample program.

Explanations

- **Checking contents of the memory**

The contents of memory can be checked in the **TEACH IN** mode by using the same procedure as in **EDIT** mode.



- **Registering a position with compensation**

When a value is keyed in after keying in address [X] or [Z], then the [INSERT] key is pressed, the value keyed in for a machine position is added for registration. This operation is useful to correct a machine position by key–in operation.

- **Registering commands other than position commands**

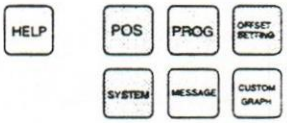
Commands to be entered before and after a machine position must be entered before and after the machine position is registered, by using the same operation as program editing in **EDIT** mode.

11 SETTING AND DISPLAYING DATA

To operate a CNC machine tool, various data must be set on the CRT/MDI panel for the CNC. The operator can monitor the state of operation with data displayed during operation.
This chapter describes how to display and set data for each function.

Explanations

• Screen transition chart



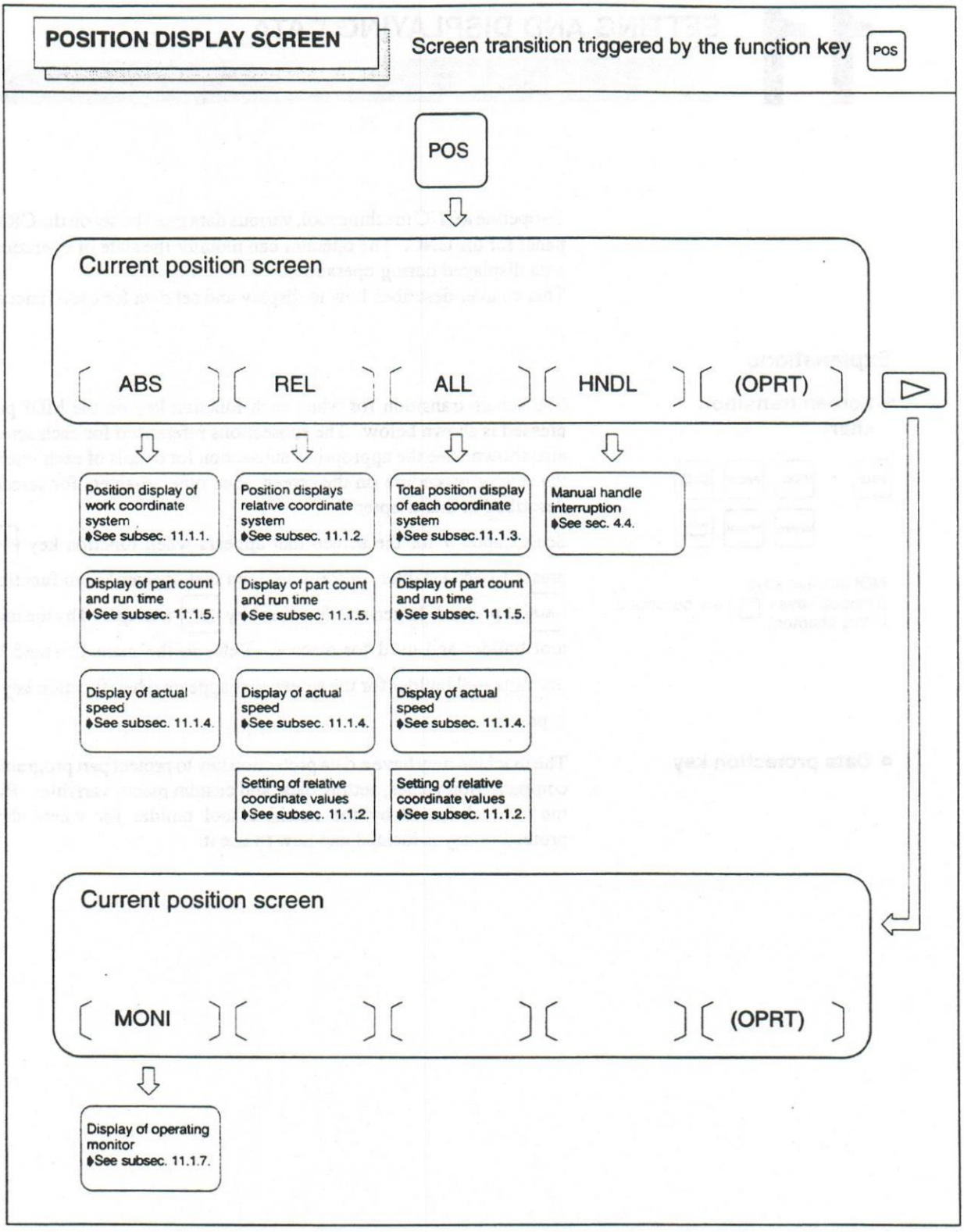
MDI function keys
(Shaded keys (▢) are described in this chapter.)

The screen transition for when each function key on the MDI panel is pressed is shown below. The subsections referenced for each screen are also shown. See the appropriate subsection for details of each screen and the setting procedure on the screen. See other chapters for screens not described in this chapter.

See Chapter 7 for the screen that appears when function key MESSAGE is pressed. See Chapter 13 for the screen that appears when function key HELP is pressed. In general, function key CUSTOM GRAPH is prepared by the machine tool builder and used for macros. Refer to the manual issued by the machine tool builder for the screen that appears when function key CUSTOM GRAPH is pressed.

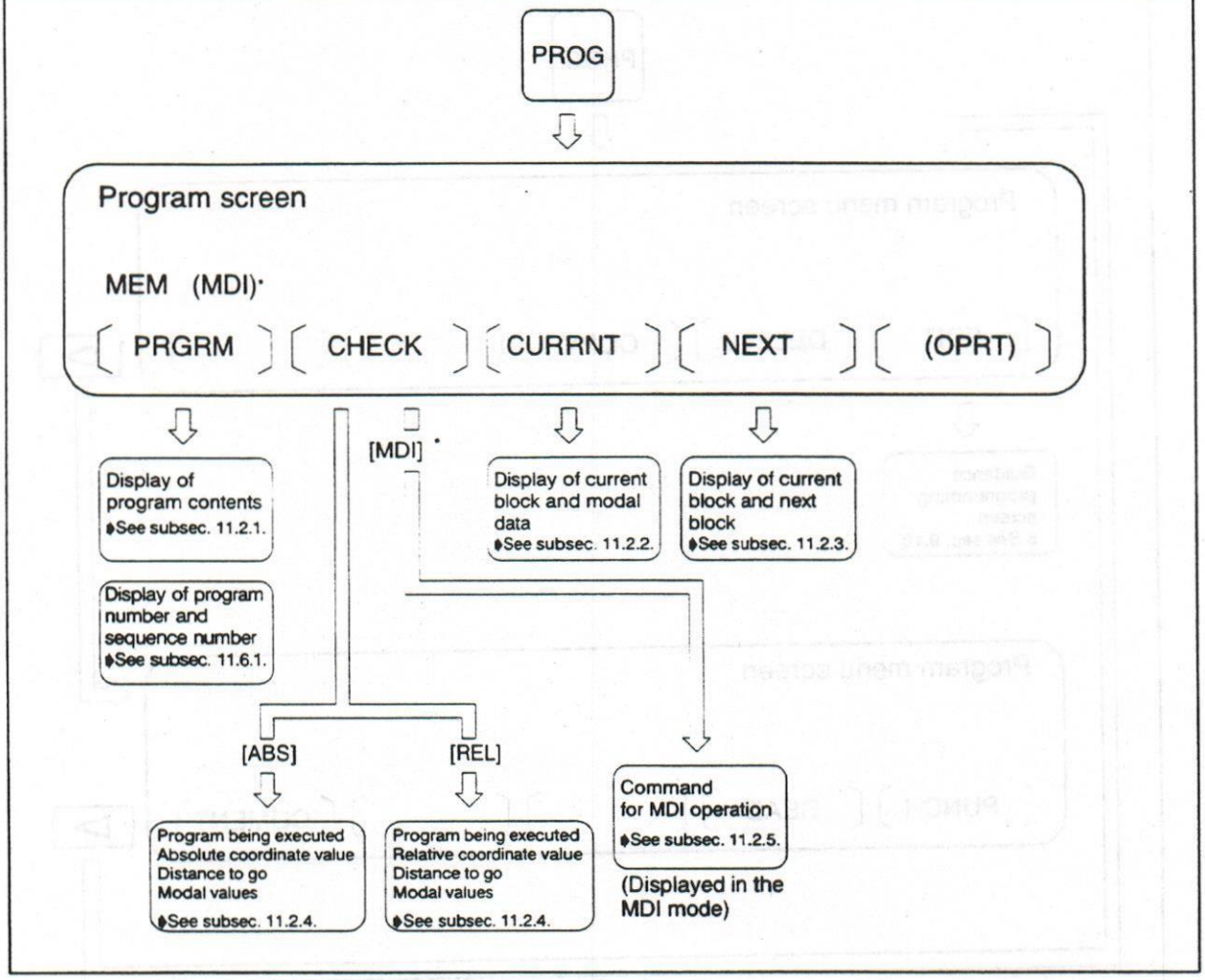
• Data protection key

The machine may have a data protection key to protect part programs, tool compensation values, setting data, and custom macro variables. Refer to the manual issued by the machine tool builder for where the data protection key is located and how to use it.


POSITION DISPLAY SCREEN
Screen transition triggered by the function key POS
POS
Current position screen
ABS
REL
ALL
HNDL
(OPRT)
Position display of work coordinate system ◊See subsec. 11.1.1.
Position displays relative coordinate system ◊See subsec. 11.1.2.
Total position display of each coordinate system ◊See subsec.11.1.3.
Manual handle interruption ◊See sec. 4.4.
Display of part count and run time ◊See subsec. 11.1.5.
Display of part count and run time ◊See subsec. 11.1.5.
Display of part count and run time ◊See subsec. 11.1.5.
Display of actual speed ◊See subsec. 11.1.4.
Display of actual speed ◊See subsec. 11.1.4.
Display of actual speed ◊See subsec. 11.1.4.
Setting of relative coordinate values ◊See subsec. 11.1.2.
Setting of relative coordinate values ◊See subsec. 11.1.2.
Current position screen
MONI
(OPRT)
Display of operating monitor ◊See subsec. 11.1.7.

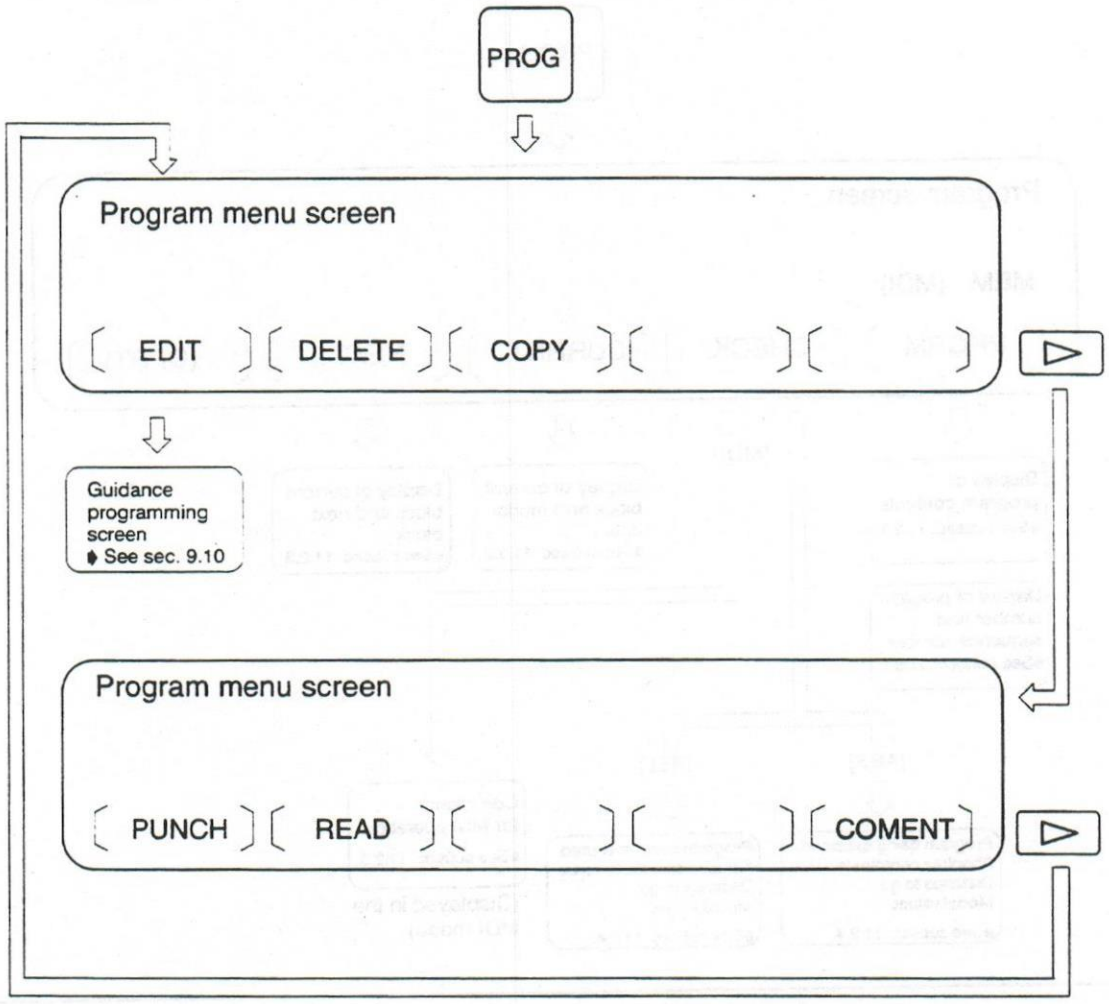
PROGRAM SCREEN

Screen transition triggered by the function key PROG in the MEMORY or MDI mode



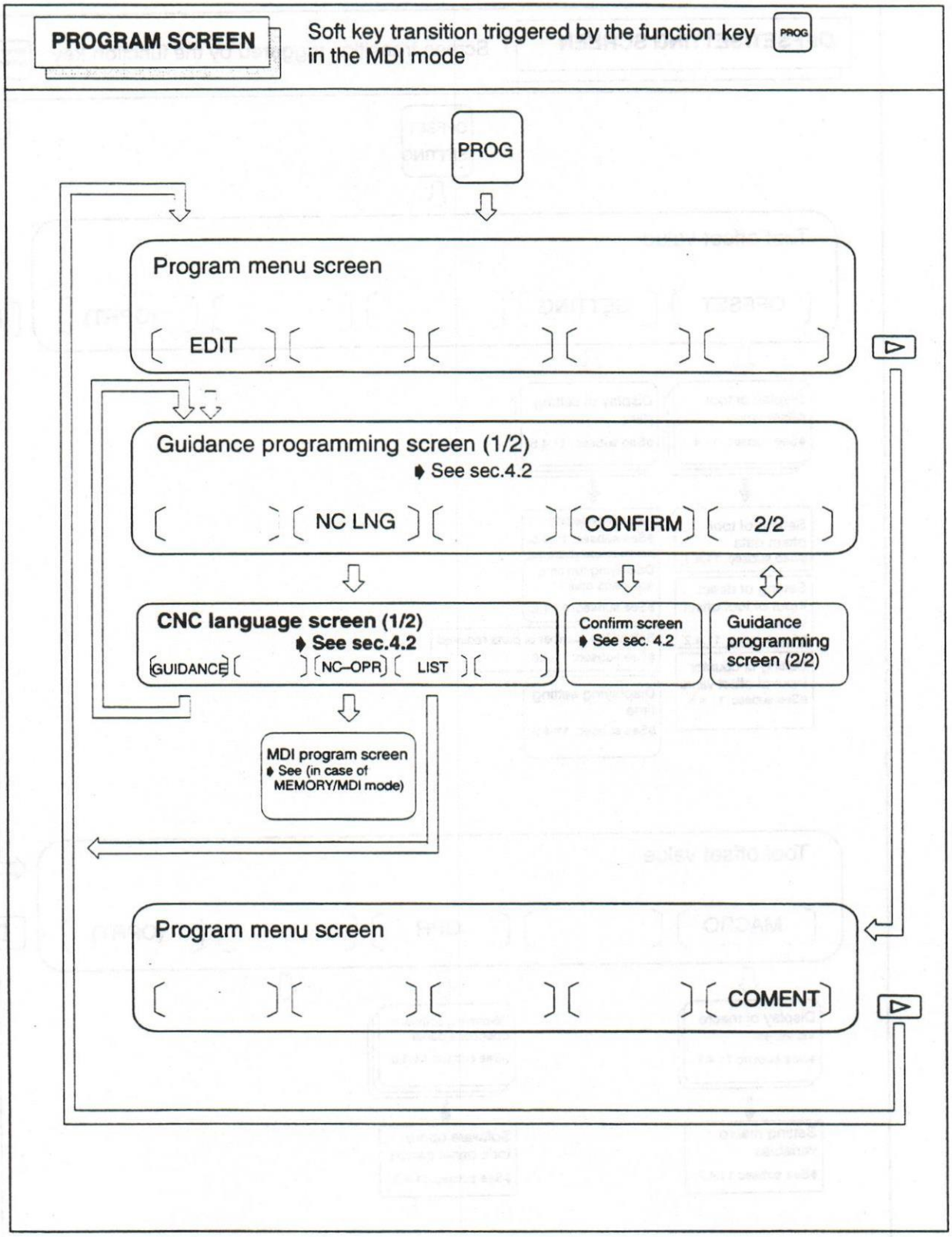
PROGRAM SCREEN

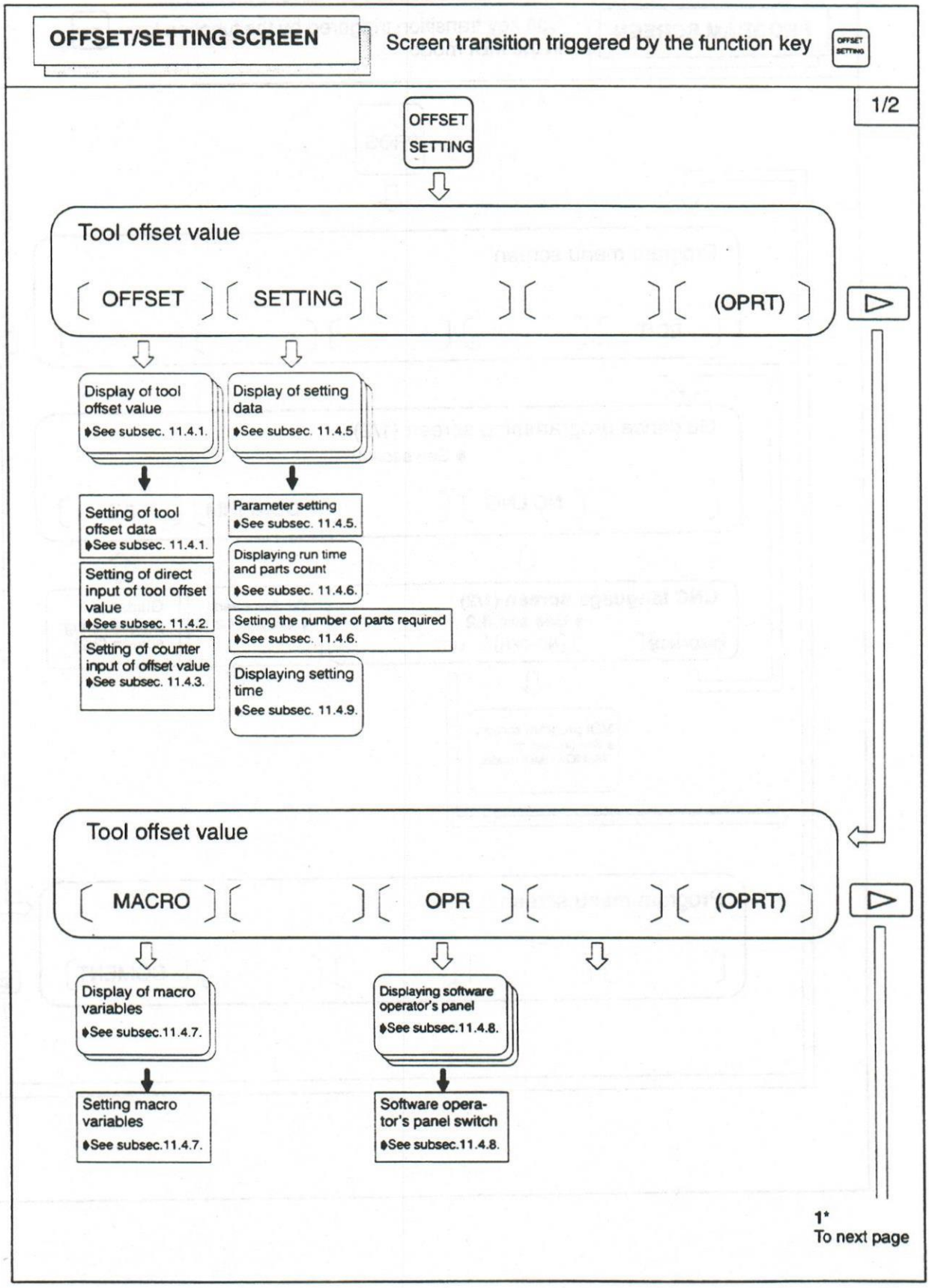
Soft key transition triggered by the function key PROG in the EDIT mode

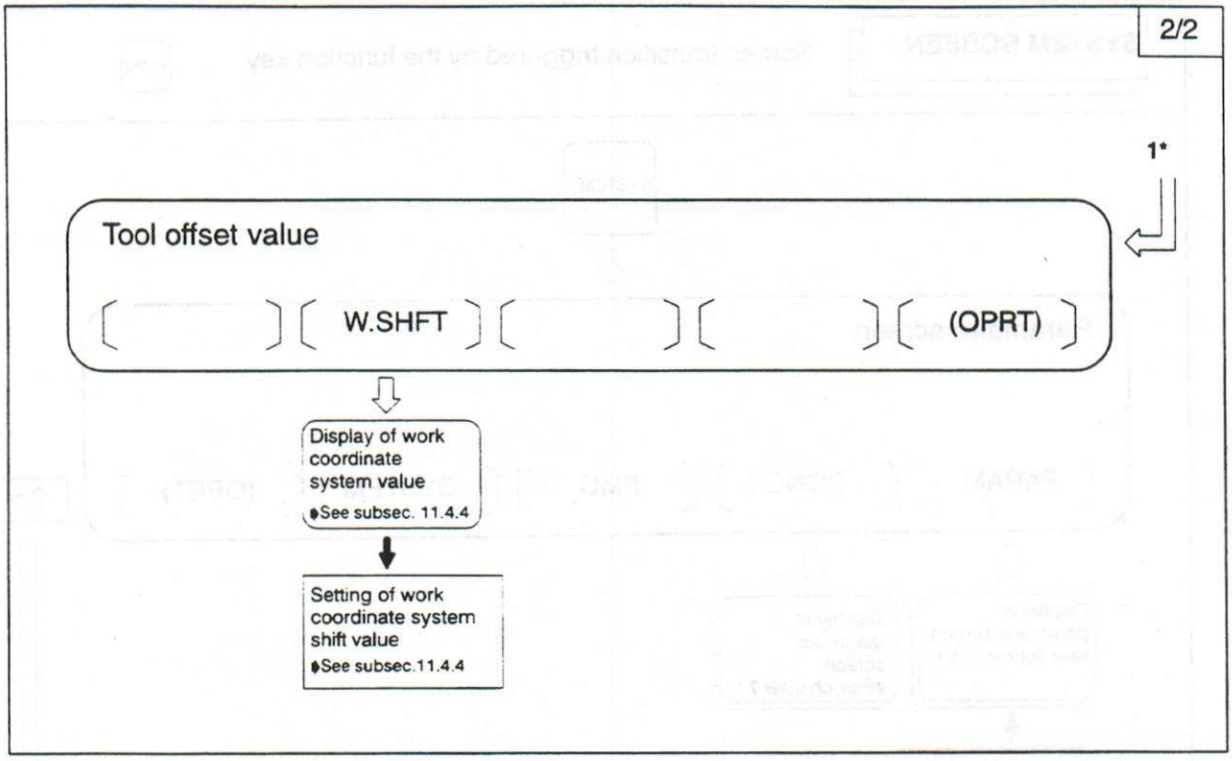

PROG
Program menu screen
[EDIT] [DELETE] [COPY] [ ] [ ] [▷]
Guidance programming screen
See sec. 9.10
Program menu screen
[PUNCH] [READ] [ ] [ ] [COMENT] [▷]

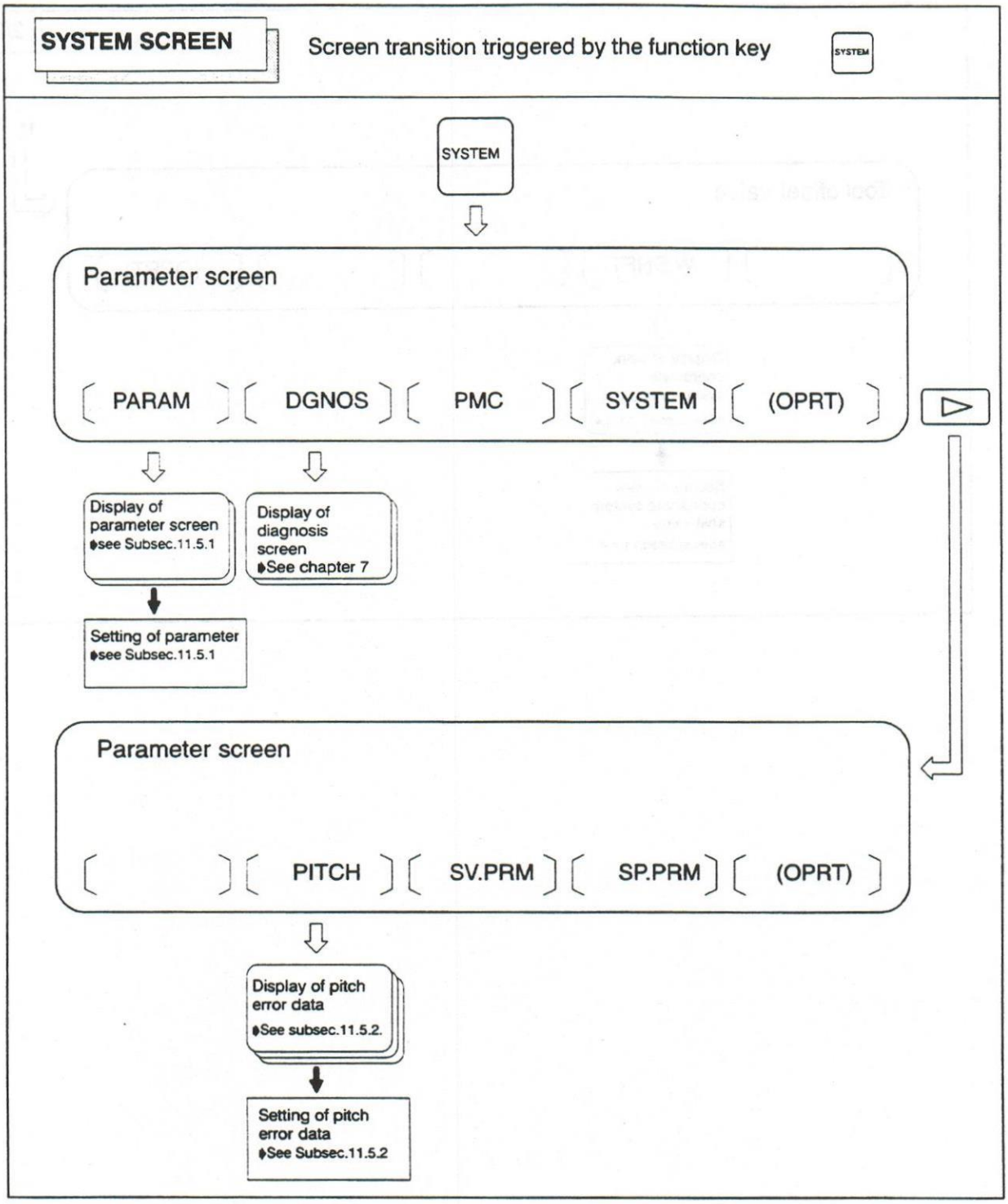
PROGRAM SCREEN

Soft key transition triggered by the function key PROG in the MDI mode


PROG
Program menu screen
[EDIT] [ ] [ ] [ ] [ ] [▷]
Guidance programming screen (1/2)
See sec.4.2
[ ] [NC LNG] [ ] [CONFIRM] [2/2]
CNC language screen (1/2)
See sec.4.2
[GUIDANCE] [ ] [NC–OPR] [LIST] [ ]
Confirm screen
See sec. 4.2
Guidance programming screen (2/2)
MDI program screen
See (in case of MEMORY/MDI mode)
Program menu screen
[ ] [ ] [ ] [ ] [COMENT] [▷]


OFFSET/SETTING SCREEN
Screen transition triggered by the function key
OFFSET SETTING
1/2
OFFSET SETTING
Tool offset value
[OFFSET] [SETTING] [ ] [ ] [(OPRT)]
Display of tool offset value
See subsec. 11.4.1.
Display of setting data
See subsec. 11.4.5
Setting of tool offset data
See subsec. 11.4.1.
Setting of direct input of tool offset value
See subsec. 11.4.2.
Setting of counter input of offset value
See subsec. 11.4.3.
Parameter setting
See subsec. 11.4.5.
Displaying run time and parts count
See subsec. 11.4.6.
Setting the number of parts required
See subsec. 11.4.6.
Displaying setting time
See subsec. 11.4.9.
Tool offset value
[MACRO] [ ] [OPR] [ ] [(OPRT)]
Display of macro variables
See subsec.11.4.7.
Setting macro variables
See subsec.11.4.7.
Displaying software operator's panel
See subsec.11.4.8.
Software operator's panel switch
See subsec.11.4.8.
1*
To next page


2/2
1*
Tool offset value
[ ] [W.SHFT] [ ] [ ] [(OPRT)]
Display of work coordinate system value
•See subsec. 11.4.4
Setting of work coordinate system shift value
•See subsec.11.4.4


SYSTEM SCREEN
Screen transition triggered by the function key
SYSTEM
SYSTEM
Parameter screen
[PARAM] [DGNOS] [PMC] [SYSTEM] [(OPRT)]
Display of parameter screen
see Subsec.11.5.1
Display of diagnosis screen
See chapter 7
Setting of parameter
see Subsec.11.5.1
Parameter screen
[ ] [PITCH] [SV.PRM] [SP.PRM] [(OPRT)]
Display of pitch error data
See subsec.11.5.2.
Setting of pitch error data
See Subsec.11.5.2

- **Setting screens**

The table below lists the data set on each screen.

Table.11. Setting screens and data on them

No.	Setting screen	Contents of setting	Reference item
1	Tool offset value	Tool offset value Tool nose radius compensation value Imaginary tool nose number	Subsec. 11.4.1
		Direct input of tool offset value	Subsec. 11.4.2
		Counter input of offset value	Subsec. 11.4.3
2	Workpiece coordinate system setting	Workpiece coordinate system shift value	Subsec. 11.4.4
3	Setting data (handy)	Parameter write TV check Punch code (EIA/ISO) Input unit (mm/inch) I/O channel Automatic insert of Sequence No.	Subsec. 11.4.5
4	Setting data (mirror image)	Mirror image	Subsec. 11.4.5
5	Setting data (timer)	Parts required	Subsec. 11.4.6
6	Macro variables	Custom macro common variables (#100A#149) (#500A#531)	Subsec. 11.4.7
7	Parameter	Parameter	Subsec. 11.5.1
8	Pitch error	Pitch error compensation data	Subsec. 11.5.2
9	software operator's panel	Mode selection Jog feed axis selection Jog rapid traverse Axis selection for Manual pulse generator Multiplication for manual pulse generator Jog feedrate Feedrate override Rapid traverse override Optional block skip Single block Machine lock Dry run Protect key Feed hold	Subsec. 11.4.8

11.1 SCREENS DISPLAYED BY FUNCTION KEY [POS]

Press function key [POS] to display the current position of the tool.

The following three screens are used to display the current position of the tool:

- **Position display screen for the work coordinate system.**
- **Position display screen for the relative coordinate system.**
- **Overall position display screen.**

The above screens can also display the feedrate, run time, and the number of parts.

Function key [POS] can also be used to display the load on the servo motor and spindle motor and the rotation speed of the spindle motor (operating monitor display).

Function key [POS] can also be used to display the screen for displaying the distance moved by handle interruption. See Section 4.6 for details on this screen.

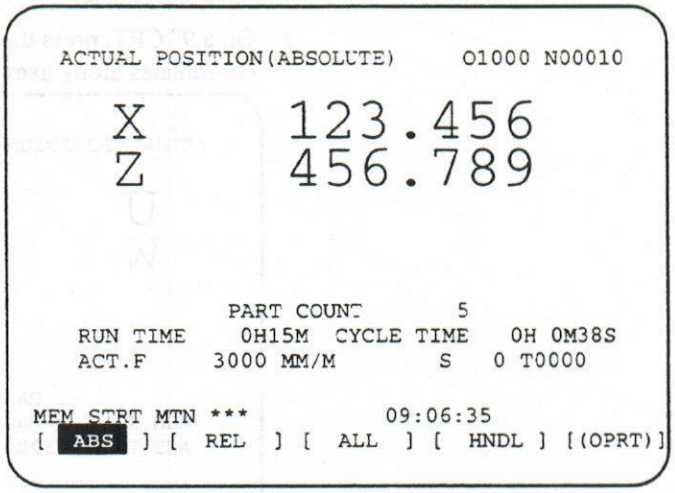
11.1.1 Position Display in the Workpiece Coordinate System

Displays the current position of the tool in the workpiece coordinate system. The current position changes as the tool moves. The least input increment is used as the unit for numeric values. The title at the top of the screen indicates that absolute coordinates are used.

Display procedure for the current position screen in the workpiece coordinate system

1 Press function key [POS].

2 Press soft key **[ABS]**.


ACTUAL POSITION(ABSOLUTE) O1000 N00010
X 123.456
Z 456.789
PART COUNT 5
RUN TIME 0H15M CYCLE TIME 0H 0M38S
ACT.F 3000 MM/M S 0 T0000
MEM STRT MTN *** 09:06:35
[ABS] [REL] [ALL] [HNDL] [(OPRT)]

Explanations

- **Display including compensation values**

Bits 6 and 7 of parameter 3104 can be used to select whether the displayed values include tool offset value and tool nose radius compensation.

11.1.2 Position Display in the Relative Coordinate System

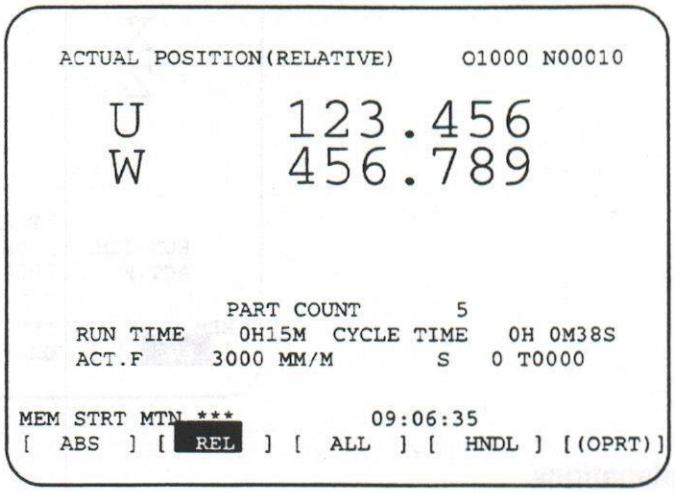
Displays the current position of the tool in a relative coordinate system based on the coordinates set by the operator. The current position changes as the tool moves. The increment system is used as the unit for numeric values. The title at the top of the screen indicates that relative coordinates are used.

Display procedure for the current position screen with the relative coordinate system

1 Press function key [POS].

2 Press soft key **[REL]**.

3 On a 9" CRT, press the **[REL]** soft key one more time to display the coordinates along axes other than the six standard axes.

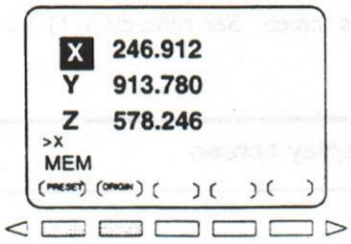

ACTUAL POSITION(RELATIVE) O1000 N00010
U 123.456
W 456.789
PART COUNT 5
RUN TIME 0H15M CYCLE TIME 0H 0M38S
ACT.F 3000 MM/M S 0 T0000
MEM STRT MTN ***
09:06:35
[ABS] [REL] [ALL] [HNDL] [(OPRT)]

Explanations

- **Setting the relative coordinates**

The current position of the tool in the relative coordinate system can be reset to 0 or preset to a specified value as follows:

Procedure to set the axis coordinate to a specified value

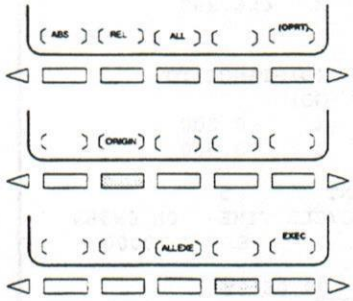


1 Enter an axis address (such as X or Z) on the screen for the relative coordinates. The indication for the specified axis blinks and the soft keys change as shown on the left.

2 · To reset the coordinate to 0, press soft key **[ORIGIN]**. The relative coordinate for the blinking axis is reset to 0.

· To preset the coordinate to a specified value, enter the value and press soft key **[PRESET]**. The relative coordinate for the blinking axis is set to the entered value.

Procedure to reset all axes



1 Press soft key **[(OPRT)]**.

2 Press soft key **[ORIGIN]**.

3 Press soft key **[ALLEXE]**.
The relative coordinates for all axes are reset to 0.

- **Display including compensation values**

Bits 4 and 5 of parameter 3104 can be used to select whether the displayed values include tool offset and tool nose radius compensation.

- **Presetting by setting a coordinate system**

Bit 3 of parameter 3104 is used to specify whether the displayed positions in the relative coordinate system are preset to the same values as in the workpiece coordinate system when a coordinate system is set by a G50 (G code system A) or a G92 (G code system B or C) command or when the manual reference position return is made.

11.1.3 Overall Position Display

Displays the following positions on a screen : Current positions of the tool in the workpiece coordinate system, relative coordinate system, and machine coordinate system, and the remaining distance. The relative coordinates can also be set on this screen. See subsection 11.1.2 for the procedure.

Procedure for displaying overall position display screen

1 Press function key [POS] .

2 Press soft key **[ALL]**.

```
ACTUAL POSITION                O1000 N00010
  (RELATIVE)                    (ABSOLUTE)
U    246.912                  X    123.456
W    913.780                  Z    456.890

  (MACHINE)                     (DISTANCE TO
X                               GO)
0.000                         X      0.000
Z                             Z      0.000
0.000
                PART COUNT          5
RUN TIME     0H15M  CYCLE TIME     0H 0M38S
ACT.F     3000 MM/M              S    0 T0000

MEM **** *** ***              09:06:35
[ ABS ] [ REL ] [ ALL ] [ HNDL ] [(OPRT)]
```

Explanations

- **Coordinate display**

The current positions of the tool in the following coordinate systems are displayed at the same time:

·Current position in the relative coordinate system (relative coordinate)
·Current position in the work coordinate system (absolute coordinate)
·Current position in the machine coordinate system (machine coordinate)
·Distance to go (distance to go)

- **Distance to go**

The distance remaining is displayed in the MEMORY or MDI mode. The distance the tool is yet to be moved in the current block is displayed.

- **Machine coordinate system**

The least command increment is used as the unit for values displayed in the machine coordinate system. However, the least input increment can be used by setting bit 0 (MCN) of parameter 3104.

- **Resetting relative coordinates**

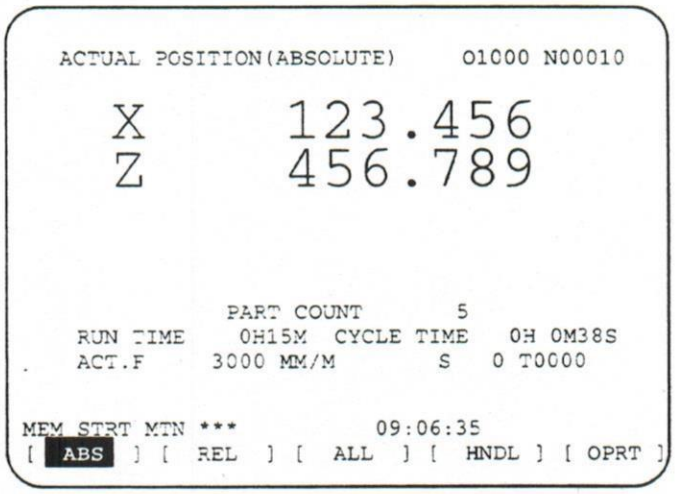
On the overall position display screen, relative coordinates can be reset to 0 or preset to specified values. The procedure is the same as that for resetting the relative coordinates described in 11.1.2.

11.1.4 Actual Feedrate Display

The actual feedrate on the machine (per minute) can be displayed on a current position display screen or program check screen by setting bit 0 (DPF) of parameter 3105.

Display procedure for the actual feedrate on the current position display screen

1 Press function key [POS] to display a current position display screen.


ACTUAL POSITION(ABSOLUTE) O1000 N00010
X 123.456
Z 456.789
PART COUNT 5
RUN TIME 0H15M CYCLE TIME 0H 0M38S
ACT.F 3000 MM/M S 0 T0000
MEM STRT MTN *** 09:06:35
[ABS] [REL] [ALL] [HNDL] [OPRT]

Actual feedrate is displayed after ACT.F.

The actual feedrate is displayed in units of millimeter/min or inch/min (depending on the specified least input increment) under the display of the current position.

Explanations

- **Actual feedrate value**

The actual rate is calculated by the following expression:

$$Fact = \sqrt{\sum_{i=1}^{n}(fi)^2}$$

where
n : Number of axes
fi : Cutting feed rate in the tangential direction of each axis or rapid traverse rate
Fact : Actual feedrate displayed

The display unit: mm/min (metric input).
inch/min (Inch input, Two digits below the decimal point are displayed.)

- **Actual feedrate display of feed per revolution**

  In the case of feed per revolution and thread cutting, the actual feedrate displayed is the feed per minute rather than feed per revolution.

- **Actual feedrate display on the other screen**

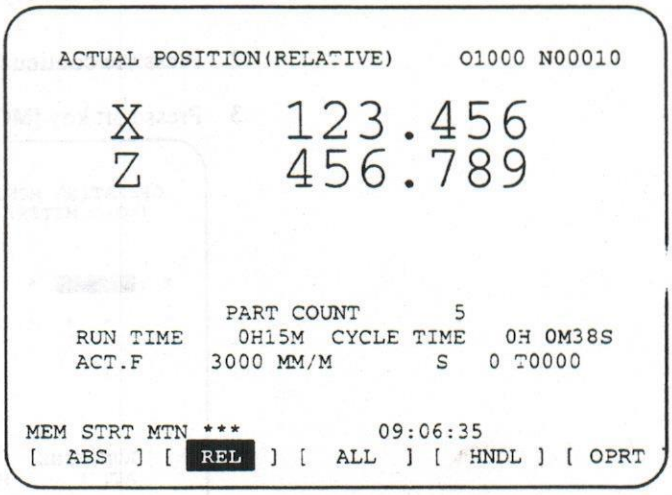
  The program check screen also displays the actual feedrate.

11.1.5 Display of Run Time and Parts Count

The run time, cycle time, and the number of machined parts are displayed on the current position display screens.

Procedure for displaying run time and parts count on the current position display screen

1 Press function key [POS] to display a current position display screen.


ACTUAL POSITION(RELATIVE) O1000 N00010
X 123.456
Z 456.789
PART COUNT 5
RUN TIME 0H15M CYCLE TIME 0H 0M38S
ACT.F 3000 MM/M S 0 T0000
MEM STRT MTN ***
09:06:35
[ABS] [REL] [ALL] [HNDL] [OPRT

The number of machined parts (PART COUNT), run time (RUN TIME), and cycle time (CYCLE TIME) are displayed under the current position.

Explanations

- **PART COUNT**

  Indicates the number of machined parts. The number is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed.

- **RUN TIME**

  Indicates the total run time during automatic operation, excluding the stop and feed hold time.

- **CYCLE TIME**

  Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- **Display on the other screen**

  Details of the run time and the number of machined parts are displayed on the setting screen. See subsection 11.4.6.

- **Parameter setting**

  The number of machined parts and run time cannot be set on current position display screens. They can be set by parameters No. 6711, 6751, and 6752 or on the setting screen.

- **Incrementing the number of machined parts**

  Bit 0 (PCM) of parameter No. 6700 is used to specify whether the number of machined parts is incremented each time M02, M30, or an M code specified by parameter No. 6710 is executed, or only each time an M code specified by parameter No. 6710 is executed.

11.1.6 Operating Monitor Display

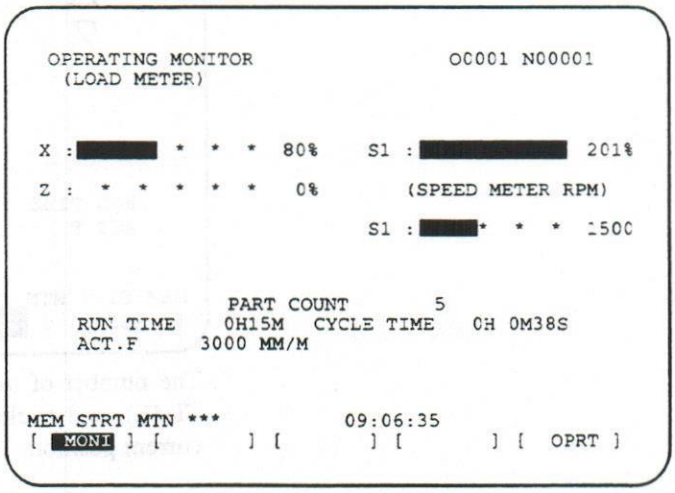
The reading on the load meter can be displayed for each servo axis and the serial spindle by setting bit 5 (OPM) of parameter 3111 to 1. The reading on the speedometer can also be displayed for the serial spindle.

Procedure for displaying the operating monitor

1 Press function key [POS] to display a current position display screen.

2 Press the continuous–menu key [▷].

3 Press soft key **[MONI]**.


OPERATING MONITOR
(LOAD METER)
O0001 N00001
X : * * * 80%
S1 : 201%
Z : * * * * * 0%
(SPEED METER RPM)
S1 : * * * 1500
PART COUNT 5
RUN TIME 0H15M CYCLE TIME 0H 0M38S
ACT.F 3000 MM/M
MEM STRT MTN ***
09:06:35
[MONI] [ ] [ ] [ ] [OPRT]

Explanations

- **Display of the servo axes**

  The reading on the load meter can be displayed for up to two servo axes by setting parameters 3151, 3152.

- **Display of the spindle axes**

  When serial spindles are used, the reading on the load meter and speedometer can be displayed only for the main serial spindle.

- **Unit of graph**

  The bar graph for the load meter shows load up to 200% (only a value is displayed for load exceeding 200%). The bar graph for the speedometer shows the ratio of the current spindle speed to the maximum spindle speed (100%).

- **Load meter**

  The reading on the load meter depends on servo parameter 2086 and spindle parameter 4127.

- **Speed meter**

  Although the speedometer normally indicates the speed of the spindle motor, it can also be used to indicate the speed of the spindle by setting bit 6 (OPS) of parameter 3111 to 1.

11.2 SCREENS DISPLAYED BY FUNCTION KEY PROG (IN MEMORY MODE OR MDI MODE)

This section describes the screens displayed by pressing function key PROG in MEMORY or MDI mode. The first four of the following screens display the execution state for the program currently being executed in MEMORY or MDI mode and the last screen displays the command values for MDI operation in the MDI mode:

1. Program contents display screen
2. Current block display screen
3. Next block display screen
4. Program check screen
5. Program screen for MDI operation
6. Operation time Stamp
7. Displaying the B–axis operation state

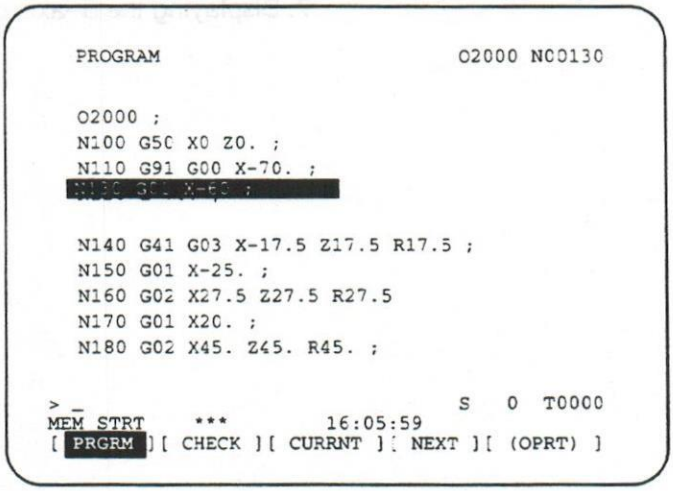
11.2.1 Program Contents Display

Displays the program currently being executed in MEMORY or MDI mode.

Procedure for displaying the program contents

1 Press function key PROG to display a program screen.

2 Press chapter selection soft key **[PRGRM]**.
The cursor is positioned at the block currently being executed.



11.2.2 Current Block Display Screen

Displays the block currently being executed and modal data in the MEMORY or MDI mode.

Procedure for displaying the current block display screen

1 Press function key PROG.

2 Press chapter selection soft key **[CURRNT]**.
The block currently being executed and modal data are displayed.
The screen displays up to 22 modal G codes and up to 11 G codes specified in the current block.

```
PROGRAM                                  O2000 N00130

        (CURRNT)          (MODAL)

   GC1 -X    100.500  G18   G00   F
       -F     50.000  G50.  G97
                      G13.  G69
                            G99
                            G21   T
                            G40   S
                            G25
                            G22
                            G80
                            G67   SACT      0
                            G54
> _                                 S    0  T0000
MEM STRT    ***       16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

11.2.3 Next Block Display Screen

Displays the block currently being executed and the block to be executed next in the MEMORY or MDI mode.

Procedure for displaying the next block display screen

1 Press function key PROG .

2 Press chapter selection soft key **[NEXT]**.
The block currently being executed and the block to be executed next are displayed.
The screen displays up to 11 G codes specified in the current block and up to 11 G codes specified in the next block.

```
PROGRAM                                  O2000 N00130

      (CURRNT)                 (NEXT)
 G00  X    17.500    G01    Z     -10.0
      Z     50.0     G98    F      1000









>_                                    S    0  T0000
MEM STRT    ***        16:05:59
[ PRGRM ][ CHECK ][ CURRNT ][ NEXT ][ (OPRT) ]
```

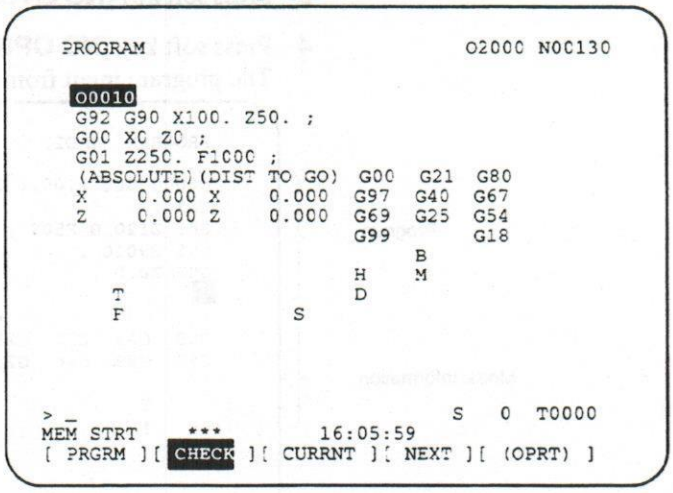
11.2.4 Program Check Screen

Displays the program currently being executed, current position of the tool, and modal data in the MEMORY mode.

Procedure for displaying the program check screen

1 Press function key [PROG].

2 Press chapter selection soft key **[CHECK]**.
The program currently being executed, current position of the tool, and modal data are displayed.



Explanations

- **Program display**

  The screen displays up to four blocks of the current program, starting from the block currently being executed. The block currently being executed is displayed in reverse video. During DNC operation, however, only three blocks can be displayed.

- **Current position display**

  The position in the workpiece coordinate system or relative coordinate system and the remaining distance are displayed. The absolute positions and relative positions are switched by soft keys **[ABS]** and **[REL]**.

- **Modal G codes**

  Up to 12 modal G codes are displayed.

- **Displaying during automatic operation**

  During automatic operation, the actual speed, SCAT, and repeat count are displayed. The key input prompt (> _) is displayed otherwise.

11.2.5 Program Screen for MDI Operation

Displays the program input from the MDI and modal data in the **MDI** mode.

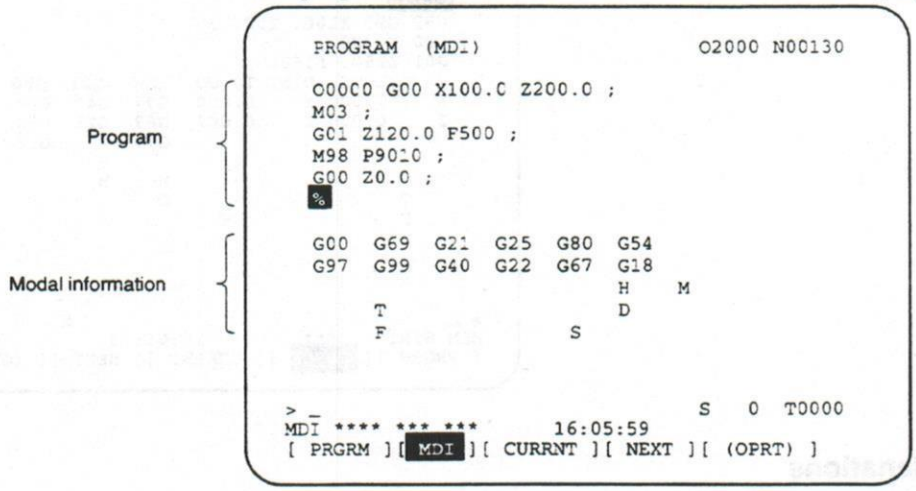
Procedure for displaying the program screen for MDI operation

1 Press function key PROG .

2 Press soft key **[EDIT]** .

3 Press soft key **[NC LNG]** . Then CNC language screen is displayed.

4 Press soft key **[NC OPR]** .
The program input from the MDI and modal data are displayed.



Explanations

- **MDI operation**

  See Section 4.2 for MDI operation.

- **Modal information**

  The modal data is displayed when bit 7 (MDL) of parameter 3107 is set to 1. Up to 16 modal G codes are displayed.

- **Displaying during automatic operation**

  During automatic operation, the actual speed and SCAT, are displayed. The key input prompt (> _) is displayed otherwise.

11.3 SCREENS DISPLAYED BY FUNCTION KEY [PROG] (IN THE EDIT MODE)

This section describes the screens displayed by pressing function key [PROG] in the EDIT mode. Function key [PROG] in the EDIT mode can display the program editing screen and the program list screen (displays memory used and a list of programs).
See Chapter 9 to for the program editing screen.

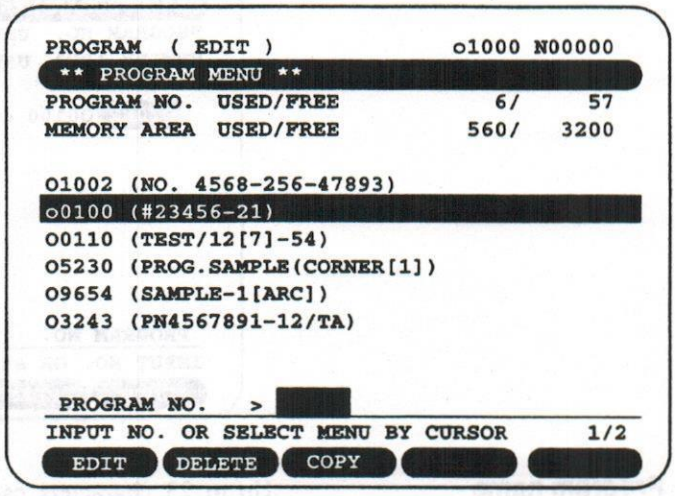
11.3.1 Displaying Memory Used and a List of Programs

Displays the number of registered programs, memory used, and a list of registered programs.

Procedure for displaying memory used and a list of programs

1 Select the **EDIT** mode.

2 Press function key [PROG].



Explanations

- **Details of memory used**

PROGRAM NO. USED

PROGRAM NO. USED : The number of the programs registered (including the subprograms)

FREE : The number of programs which can be registered additionally.

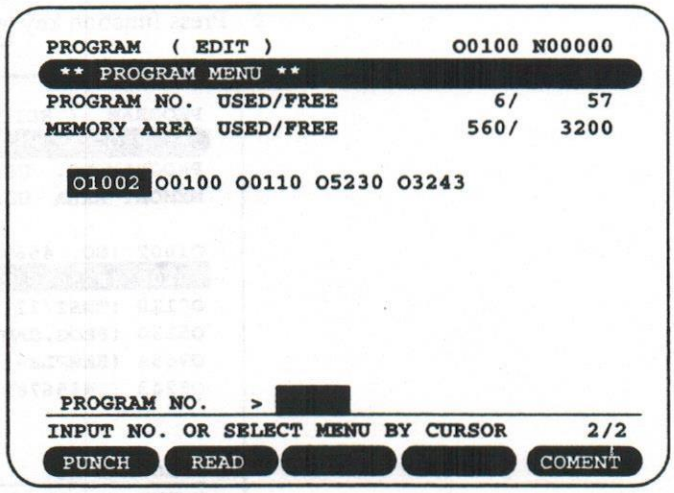
MEMORY AREA USED

MEMORY AREA USED : The capacity of the program memory in which data is registered (indicated by the number of characters).

FREE : The capacity of the program memory which can be used additionally (indicated by the number of characters).

- **Program library list**

Program Nos. registered are indicated.
Pressing the **[COMENT]** soft key on the program list screen hides the program names on the screen.



- **Program name**

Up to 23 characters can be used for naming a program within the parentheses.
Only program number is displayed for the program without any program name.

O ¢¢¢¢ (ΔΔΔΔ...Δ) ;

Program number — Program name (up to 23 characters)

- **Order in which programs are displayed in the program library list**

Programs are displayed in the same order that they are registered in the program library list. However, if bit 4 (SOR) of parameter 3107 is set to 1, programs are displayed in the order of program number starting from the smallest one.

- **Order in which programs are registered**

In the no program registered state, each program is registered after the last program in the list.
If some programs in the list were deleted, then a new program is registered, the new program is inserted in the empty location in the list created by the deleted programs.

Example) When bit 4 (SOR) of parameter 3107 is 0

1. **After clearing all programs, register programs O0001, O0002, O0003, O0004, and O0005 in this order. The program library list displays the programs in the following order:**
   O0001, O0002, O0003, O0004, O0005

2. **Delete O0002 and O0004. The program library list displays the programs in the following order:O0001, O0003, O0005**

3. **Register O0009. The program library list displays the programs in the following order:**
   O0001, O0009, O0003, O0005

11.4 SCREENS DISPLAYED BY FUNCTION KEY [OFFSET SETTING]

Press function key [OFFSET SETTING] to display or set tool compensation values and other data.
This section describes how to display or set the following data:

1. Tool offset value
2. Settings
3. Run time and part count
4. Workpiece coordinate system shift value
5. Custom macro common variables
6. Software operator's panel

This section also describes following functions.

- Direct input of tool offset value
- Counter input of offset value
- Direct input of workpiece coordinate system shift

The following functions depend on the specifications of the machine tool builder. See the manual issued by the machine tool builder for details.

- Direct input of tool offset value
- Software operator's panel

11.4.1 Displaying and Setting the Tool Offset Value

Dedicated screens are provided for displaying and setting tool offset values and tool nose radius compensation values.

Procedure for setting and displaying the tool offset value and the tool nose radius compensation value

1 Press function key [OFFSET SETTING].

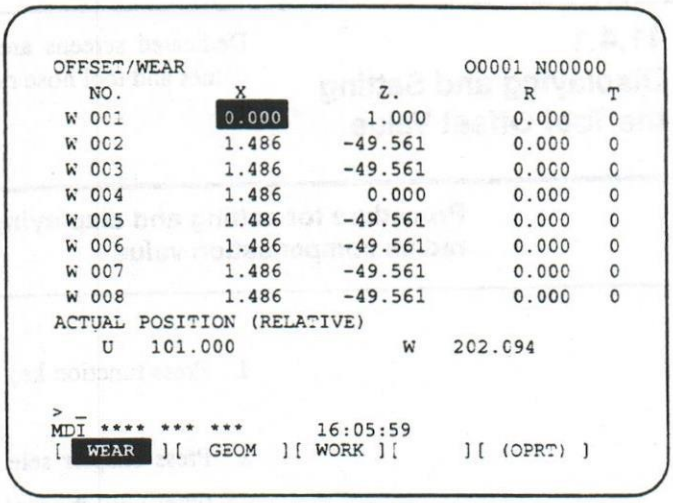
2 Press chapter selection soft key **[OFFSET]** or press [OFFSET SETTING] several times until the tool compensation screen is displayed.
Different screens are displayed depending on whether tool geometry offset, wear offset, or neither is applied.

```
OFFSET                                  O0001 N00000
   NO.          X              Z.            R      T
   001      0.000          10.000         0.000    0
   002        0.000         0.000         0.000    0
   003        0.000         0.000         0.000    0
   004       40.000       -40.000         0.000    0
   005        0.000         0.000         0.000    0
   006        0.000         0.000         0.000    0
   007        0.000         0.000         0.000    0
   008        0.000         0.000         0.000    0
ACTUAL POSITION (RELATIVE)
    U   101.000                W   202.094

>_
MDI **** *** ***       16:05:59
[ OFFSET ][ SETING ][       ][       ][ (OPRT) ]
```

Without tool geometry/wear offset

```
OFFSET/GEOMETRY                         O0001 N00000
   NO.          X              Z.            R      T
 G 001      0.000          1.000         0.000    0
 G 002        1.486      -49.561         0.000    0
 G 003        1.486      -49.561         0.000    0
 G 004        1.486        0.000         0.000    0
 G 005        1.486      -49.561         0.000    0
 G 006        1.486      -49.561         0.000    0
 G 007        1.486      -49.561         0.000    0
 G 008        1.486      -49.561         0.000    0
ACTUAL POSITION (RELATIVE)
    U   101.000                W   202.094

>_
MDI **** *** ***       16:05:59
[ WEAR ][ GEOM ][ WORK ][       ][ (OPRT) ]
```

With tool geometry offset



With tool wear offset

3 Move the cursor to the compensation value to be set or changed using page keys and cursor keys, or enter the compensation number for the compensation value to be set or changed and press soft key **[NO.SRH]**.

4 To set a compensation value, enter a value and press soft key **[INPUT]**. To change the compensation value, enter a value to add to the current value (a negative value to reduce the current value) and press soft key **[+INPUT]**. Or, enter a new value and press soft key **[INPUT]**.
TIP is the number of the virtual tool tip (see Programming).
TIP may be specified on the geometry compensation screen or on the wear compensation screen.

Explanations

- **Decimal point input**

  A decimal point can be used when entering a compensation value.

- **Other method**

  An external input/output device can be used to input or output a cutter compensation value. See Chapter 8.
  Tool compensation values can be set using the following functions described in subsequent subsections: direct input of tool offset value and counter input of offset value.

- **Tool offset memory**

  16 groups are provided for tool compensation.

- **Disabling entry of compensation values**

In some cases, tool wear compensation or tool geometry compensation values cannot be input because of the settings in bits 0 (WOF) and 1 (GOF) of parameter 3290. The input of tool compensation values from the MDI can be inhibited for a specified range of offset numbers. The first offset number for which the input of a value is inhibited is set in parameter No. 3294. The number of offset numbers, starting from the specified first number, for which the input of a value is inhibited is set in parameter No. 3295.

Consecutive input values are set as follows:

1) When values are input for offset numbers, starting from one for which input is not inhibited to one for which input is inhibited, a warning is issued and values are set only for those offset numbers for which input is not inhibited.
2) When values are input for offset numbers, starting from one for which input is inhibited to one for which input is not inhibited, a warning is issued and no values are set.

- **Displaying radius and TIP**

The radius and TIP are not added if the tool tip radius compensation option is not displayed.

- **Changing offset values during automatic operation**

When offset values have been changed during automatic operation, bit 4 (LGT) and bit 6 (LWM) of parameter 5002 can be used for specifying whether new offset values become valid in the next move command or in the next T code command.

LGT	LWM	When geometry compensation values and wear compensation values are separately specified	When geometry compensation values and wear compensation values are not separately specified
0	0	Become valid in the next T code block	Become valid in the next T code block
1	0	Become valid in the next T code block	Become valid in the next T code block
0	1	Become valid in the next T code block	Become valid in the next move command
1	1	Become valid in the next move command	Become valid in the next move command

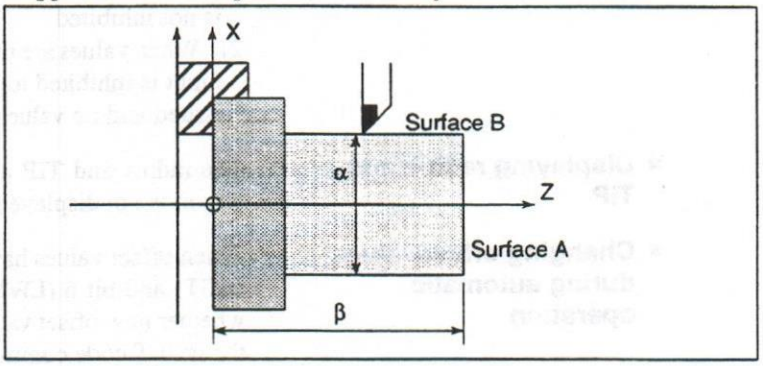
11.4.2 Direct Input of Tool Offset Value

To set the difference between the tool reference position used in programming (the nose of the standard tool, turret center, etc.) and the tool tip position of a tool actually used as an offset value

Procedure for direct input of tool offset value

- **Setting of Z axis offset value**

1 Cut surface A in manual mode with an actual tool.
Suppose that a workpiece coordinate system has been set.



2 Release the tool in X axis direction only, without moving Z axis and stop the spindle.

3 Measure distance β from the zero point in the workpiece coordinate system to surface A.
Set this value as the measured value along the Z–axis for the desired offset number, using the following procedure:

```
OFFSET/GEOMETRY                          O0001 N00000
   NO.          X              Z.             R        T
 G 001        0.000            1.000          0.000    0
 G 002        1.486          -49.561          0.000    0
 G 003        1.486          -49.561          0.000    0
 G 004        1.486            0.000          0.000    0
 G 005        1.486          -49.561          0.000    0
 G 006        1.486          -49.561          0.000    0
 G 007        1.486          -49.561          0.000    0
 G 008        1.486          -49.561          0.000    0
ACTUAL POSITION (RELATIVE)
    U      0.000                W      0.000

>Z120._
MDI **** *** ***        16:05:59
[NO,SRH][ MEASUR ][ INP.C. ][ +INPUT ][ INPUT ]
```

3–1 Press the function key [OFFSET SETTING] or the soft key **[OFFSET]** to display the tool compensation screen. If geometry compensation values and wear compensation values are separately specified, display the screen for either of them.

3–2 Move the cursor to the set offset number using cursor keys.

3–3 Press the address key [Z] to be set.

3–4 Key in the measured value (β).

3–5 Press the soft key **[MESURE]**.
The difference between measured value β and the coordinate is set as the offset value.

- **Setting of X axis offset value**

4 Cut surface B in manual mode.

5 Release the tool in the Z–axis direction without moving the X–axis and stop the spindle.

6 Measure the diameter α of surface B.
Set this value as the measured value along the X–axis for the desired offset number in the same way as when setting the value along the Z–axis.

7 Repeat above procedure the same time as the number of the necessary tools. The offset value is automatically calculated and set.
For example, in case α=69.0 when the coordinate value of surface B in the diagram above is 70.0, set 69.0 **[MEASURE]** at offset No.2.
In this case, 1.0 is set as the X–axis offset value to offset No.2.

Explanations

- **Compensation values for a program created in diameter programming**

Enter diameter values for the measured values for axes for which diameter programming is used.

- **Tool geometry offset value and tool wear offset value**

If measured values are set on the tool geometry compensation screen, all compensation values become geometry compensation values and all wear compensation values are set to 0. If measured values are set on the tool wear compensation screen, the differences between the measured compensation values and the current wear compensation values become the new compensation values.

- **Retracting along two axes**

If a record button is provided on the machine, the tool can retract along two axes when bit 2 (PRC) of parameter 5005 is set and the record signal is used. Refer to the appropriate manual issued by the machine tool builder.

11.4.3 Counter Input of Offset Value

By moving the tool until it reaches the desired reference position, the corresponding tool offset value can be set.

Procedure for counter input of offset value

1 Manually move the reference tool to the reference position.

2 Reset the relative coordinates along the axes to 0 (see subsec. 11.1.2).

3 Move the tool for which offset values are to be set to the reference position.

4 Select the tool compensation screen. Move the cursor to the offset value to be set using cursor keys.

```
OFFSET/GEOMETRY                          O0001 N00000
   NO.          X              Z.            R       T
 G 001        0.000          1.000         0.000     0
 G 002        1.486        -49.561         0.000     0
 G 003        1.486        -49.561         0.000     0
 G 004        1.486          0.000         0.000     0
 G 005        1.486        -49.561         0.000     0
 G 006        1.486        -49.561         0.000     0
 G 007        1.486        -49.561         0.000     0
 G 008        1.486        -49.561         0.000     0
ACTUAL POSITION (RELATIVE)
    U     0.000                  W      0.000


>X_
HND **** *** ***       16:05:59
[NO,SRH][ MEASUR ][ INP.C. ][ +INPUT ][ INPUT ]
```

5 Press address key [X] (or [Z]) and the soft key **[INP.C.]**.

The relative coordinates are set as the tool offset value.

Explanations

- **Geometry offset and wear offset**

When the above operations are performed on the tool geometry compensation screen, tool geometry compensation values are input and tool wear compensation values do not change.
When the above operations are performed on the tool wear compensation screen, tool wear compensation values are input and tool geometry compensation values do not change.

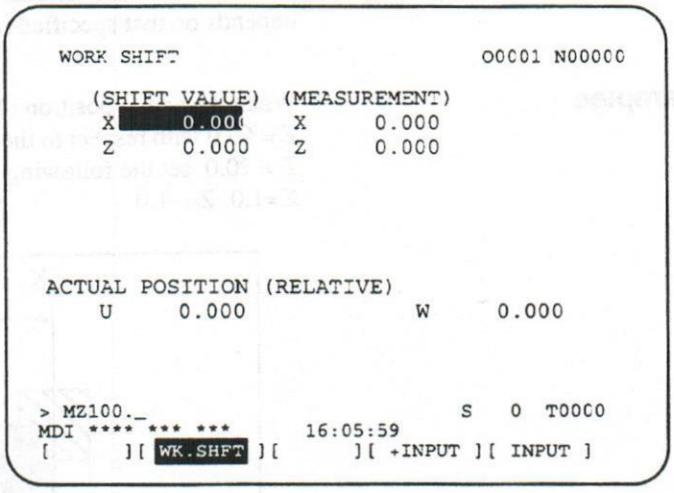
11.4.4 Setting the Workpiece Coordinate System Shifting Amount

The set coordinate system can be shifted when the coordinate system which has been set by a G50 command (or G92 command for G code system B or C) or automatic coordinate system setting is different from the workpiece coordinate system assumed at programming.

Procedure for setting the workpiece coordinate system shifting amount

1 Press function key [OFFSET SETTING].

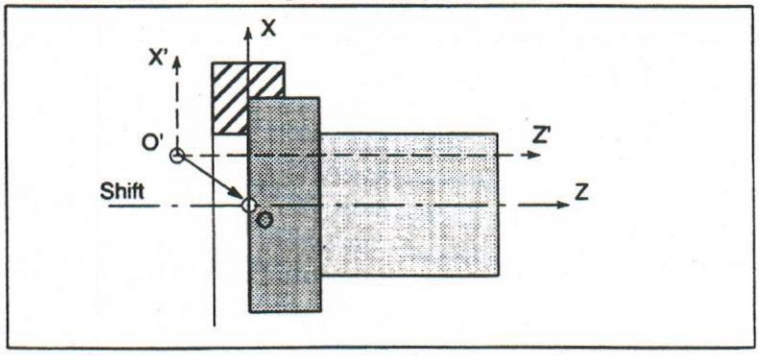
2 Press the continuous menu key [▷] several times until the screen with soft key **[WK.SHFT]** is displayed.



3 Press soft key **[WK.SHFT]**.

4 Move the cursor using cursor keys to the axis along which the coordinate system is to be shifted.

5 Enter the shift value and press soft key **[INPUT]**.



Explanations

- **When shift values become valid**

  Shift values become valid immediately after they are set.

- **Shift values and coordinate system setting command**

  Setting a command (G50 or G92) for setting a coordinate system disables the set shift values.

  Example When G50 X100.0 Z80.0; is specified, the coordinate system is set so that the current tool reference position is X = 100.0, Z = 80.0 regardless of the shift values.

- **Shift values and coordinate system setting**

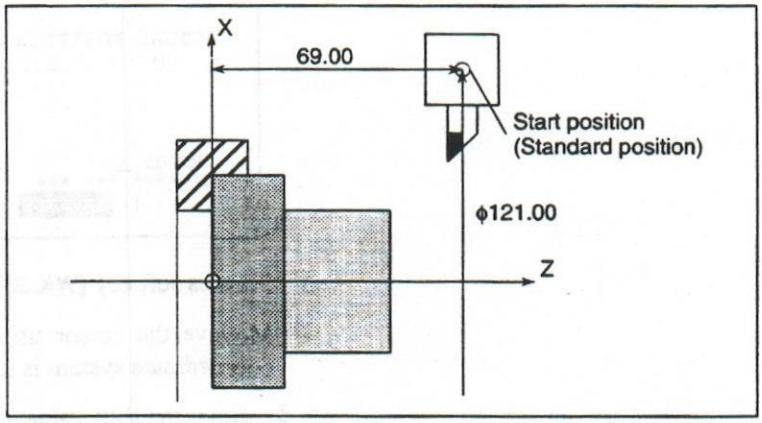
  If the automatic coordinate system setting is performed by manual reference position return after shift amount setting, the coordinate system is shifted instantly.

- **Diameter or radius value**

  Whether the shift amount on the X axis is diameter or radius value depends on that specified in program.

Examples

When the actual position of the reference point is X = 121.0 (diameter), Z = 69.0 with respect to the workpiece origin but it should be X = 120.0, Z = 70.0, set the following shift values:
X=1.0, Z=–1.0



11.4.5 Displaying and Entering Setting Data

Data such as the TV check flag and punch code is set on the setting data screen. On this screen, the operator can also enable/disable parameter writing, enable/disable the automatic insertion of sequence numbers in program editing.
See Chapter 10 for automatic insertion of sequence numbers.
This subsection describes how to set data.

Procedure for setting the setting data

1 Select the **MDI** mode.

2 Press function key [OFFSET SETTING].

3 Press soft key **[SETING]** to display the setting data screen.
This screen consists of several pages.
Press page key [PAGE ↑] or [PAGE ↓] until the desired screen is displayed.
An example of the setting data screen is shown below.

```
SETTING (HANDY)                          O0001 N00000

  PARAMETER WRITE = 1 0:DISABLE 1:ENABLE)
  TV CHECK = 0(0:OFF    1:ON)
  PUNCH CODE = 1(0:EIA    1:ISO)
  INPUT UNIT = 0(0:MM     1:INCH)
  I/O CHANNEL  =  0(0-3:CHANNEL NO.)
  SEQUENCE NO. = 0(0:OFF    1:ON)


>_
MDI **** *** ***        16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

```
SETTING (HANDY)                          O0001 N00000

  MIRROR IMAGE X=  :OFF    1:ON)
  MIRROR IMAGE Z= 0 (0:OFF    1:ON)


>_
MDI **** *** ***        16:05:59
[ OFFSET ][ SETING ][ WORK ][      ][ (OPRT) ]
```

4 Move the cursor to the item to be changed by pressing cursor keys [↑], [↓], [←], or [→].

5 Enter a new value and press soft key **[INPUT]**.

Contents of settings

- **PARAMETER WRITE**

  Setting whether parameter writing is enabled or disabled.
  0 : Disabled
  1 : Enabled

- **TV CHECK**

  Setting to perform TV check.
  0 : No TV check
  1 : Perform TV check

- **PUNCH CODE**

  Setting code when data is output through reader puncher interface.
  0 : EIA code output
  1 : ISO code output

- **INPUT UNIT**

  Setting a program input unit, inch or metric system
  0 : Metric
  1 : Inch

- **I/O CHANNEL**

  Using channel of reader/puncher interface.
  0 : Channel 0
  1 : Channel 1
  2 : Channel 2

- **SEQUENCE STOP**

  Setting of whether to perform automatic insertion of the sequence number or not at program edit in the EDIT mode.
  0 : Does not perform automatic sequence number insertion.
  1 : Perform automatic sequence number insertion.

- **MIRROR IMAGE**

  Setting of mirror image ON/OFF for each axes.
  0 : Mirror image off
  1 : Mirror image on

- **Others**

  Page key [PAGE↑] or [PAGE↓] can also be pressed to display the SETTING (TIMER) screen. See subsection 11.4.6 for this screen.

11.4.6 Displaying and Setting Run Time, Parts Count, and Time

Various run times, the total number of machined parts, number of parts required, and number of machined parts can be displayed. This data can be set by parameters or on this screen (except for the total number of machined parts and the time during which the power is on, which can be set only by parameters).
This screen can also display the clock time. The time can be set on the screen.

Procedure for Displaying and Setting Run Time, Parts Count and Time

1 Select the MDI mode.

2 Press function key [OFFSET SETTING] .

3 Press chapter selection soft key **[SETING]**.

4 Press page key [PAGE ↑] or [PAGE ↓] several times until the following screen is displayed.

```
SETTING (TIMER)                          OC001 N00000

    PARTS TOTAL  = 14
    PARTS REQUIRED=
    PARTS COUNT  = 23

    POWER ON = 4H 31M
    OPERATING TIME= 0H  0M  0S
    CUTTING TIME = 0H 37M  5S
    FREE PURPOSE = 0H  0M  0S
    CYCLE TIME = 0H  0M  0S
                 DATE =  1994/07/05
                 TIME =    11:32:52

>_
MDI **** *** ***        16:05:59
[ OFFSET ][ SETING ][      ][      ][ (OPRT) ]
```

5 To set the number of parts required, move the cursor to PARTS REQUIRED and enter the number of parts to be machined.

6 To set the clock, move the cursor to DATE or TIME, enter a new date or time, then press soft key **[INPUT]**.

Display items

- **PARTS TOTAL**

This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. This value cannot be set on this screen. Set the value in parameter 6712.

- **PARTS REQUIRED**

It is used for setting the number of machined parts required.
When the "0" is set to it, there is no limitation to the number of parts. Also, its setting can be made by the parameter (NO. 6713).

- **PARTS COUNT**

  This value is incremented by one when M02, M30, or an M code specified by parameter 6710 is executed. The value can also be set by parameter 6711. In general, this value is reset when it reaches the number of parts required. Refer to the manual issued by the machine tool builder for details.

- **POWER ON**

  Displays the total time which the power is on. This value cannot be set on this screen but can be preset in parameter 6750.

- **OPERATING TIME**

  Indicates the total run time during automatic operation, excluding the stop and feed hold time.
  This value can be preset in parameter 6751 or 6752.

- **CUTTING TIME**

  Displays the total time taken by cutting that involves cutting feed such as linear interpolation (G01) and circular interpolation (G02 or G03). This value can be preset in parameter 6753 or 6754.

- **FREE PURPOSE**

  This value can be used, for example, as the total time during which coolant flows. Refer to the manual issued by the machine tool builder for details.

- **CYCLE TIME**

  Indicates the run time of one automatic operation, excluding the stop and feed hold time. This is automatically preset to 0 when a cycle start is performed at reset state. It is preset to 0 even when power is removed.

- **DATA and TIME**

  Displays the current date and time. The date and time can be set on this screen.

Explanations

- **Usage**

  When the command of M02 or M30 is executed, the total number of machined parts and the number of machined parts are incremented by one. Therefore, create the program so that M02 or M30 is executed every time the processing of one part is completed. Furthermore, if an M code set to the parameter (NO. 6710) is executed, counting is made in the similar manner. Also, it is possible to disable counting even if M02 or M30 is executed (parameter PCM (No. 6700#0) is set to 1). For details, see the manual issued by machine tool builders.

Restrictions

- **Run time and part count settings**

  Negative value cannot be set. Also, the setting of "M" and "S" of run time is valid from 0 to 59.
  Negative value may not be set to the total number of machined parts.

- **Time settings**

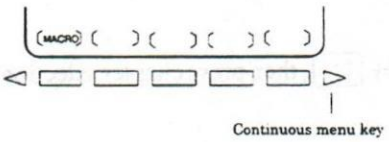
  Neither negative value nor the value exceeding the value in the following table can be set.

Item	Maximum value	Item	Maximum value
Year	2085	Hour	23
Month	12	Minute	59
Day	31	Second	59

11.4.7 Displaying and Setting Custom Macro Common Variables

Displays common variables (#100 to #149 or #100 to #199, and #500 to #531 or #500 to #999) on the CRT. When the absolute value for a common variable exceeds 99999999, ******** is displayed. The values for variables can be set on this screen. Relative coordinates can also be set to variables.

Procedure for displaying and setting custom macro common variables



1 Press function key [OFFSET SETTING].

2 Press the continuous menu key [▷], then press chapter selection soft key **[MACRO]**.The following screen is displayed:

```
VARIABLE                                  O0001 N00000
  NO.       DATA          NO.       DATA
  100   1000.000          108        0.000
  101      0.000          109    40000.000
  102 -50000.000          110    153020.00
  103      0.000          111     0001.000
  104  1238501.0          112        0.000
  105      0.000          113    20000.000
  106      0.000          114        0.000
  107      0.000          115        0.000
ACTUAL POSITION (RELATIVE)
    U      0.000            W        0.000

> _                                   S   0  T0000
MDI **** *** ***       16:05:59
[ NO.SRH ] [     ] [ INP.C. ] [     ] [ INPUT ]
```

3 Move the cursor to the variable number to set using either of the following methods:
- Enter the variable number and press soft key **[NO.SRH]**.
- Move the cursor to the variable number to set by pressing page keys [PAGE↑] and/or [PAGE↓] and cursor keys [↑], [↓], [←], and/or [→].

4 Enter data with numeric keys and press soft key **[INPUT]**.

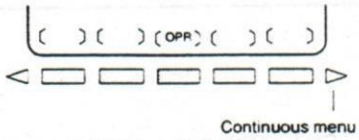
5 To set a relative coordinate in a variable, press address key [X] or [Z], then press soft key **[INP.C.]**.

6 To set a blank in a variable, just press soft key **[INPUT]**. The value field for the variable becomes blank.

11.4.8 Displaying and Setting the Software Operator's Panel

With this function, functions of the switches on the machine operator's panel can be controlled from the CRT/MDI panel.
Jog feed can be performed using numeric keys.

Procedure for displaying and setting the software operator's panel



1 Press function key [OFFSET SETTING].

2 Press the continuous menu key [▷], then press chapter selection soft key **[OPR]**.

3 The screen consists of several pages.
Press page key [PAGE ↑] or [PAGE ↓] until the desired screen is displayed.

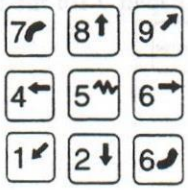
```
OPERATOR'S PANEL                    O0000 N00000

 MODE :  MDI  MEM  EDIT  HND  JOG  REF
 HANDLE AXIS :   HX   HZ
 STEP MULTI. :   *1   *10   *100
 RAPID OVRD. :  100%  50%   25%   F0
 JOG FEED  :     2.0%
                ***************
 FEED OVRD. :  100%
               ****
ACTUAL POSITION (ABSOLUTE)
    X      0.000         Z      0.000


>_
REF **** *** ***        16:05:59
[ MACRO ][        ][ OPR  ][       ][       ]
```

```
OPERATOR'S PANEL                    O0000 N00000

BLOCK SKIP        OFF      ON
SINGLE BLOCK :    OFF      ON
MACHINE LOCK :    OFF      ON
PROTECT KEY  :    PROTECT  RELEASE
FEED HOLD    :    OFF

ACTUAL POSITION (ABSOLUTE)
    X      0.000         Z      0.000



MDI **** *** ***        16:05:59
[ MACRO ][        ][ OPR  ][       ][       ]
```

4 Move the cursor to the desired switch by pressing cursor key ↑ or ↓.

5 Push the cursor move key ← or → to match the mark ■ to an arbitrary position and set the desired condition.

6 Press one of the following arrow keys to perform jog feed. Press the 5 key together with an arrow key to perform jog rapid traverse.



Explanations

- **Valid operations**

The valid operations on the software operator's panel are shown below. Whether to use the CRT or machine operator's panel for each group of operations can be selected by parameter 7200.

Group1 : Mode selection
Group2 : Selection of jog feed axis, jog rapid traverse
Group3 : Selection of manual pulse generator feed axis, selection of manual pulse magnification x1, x10, x100
Group4 : Jog federate, federate override, rapid traverse override
Group5 : Optional block skip, single block, machine lock, dry run
Group6 : Protect key
Group7 : Feed hold

- **Display**

The groups for which the machine operator's panel is selected by parameter 7200 are not displayed on the software operator's panel.

- **Screens on which jog feed is valid**

When the CRT indicates other than the software operator's panel screen and diagnostic screen, jog feed is not conducted even if the arrow key is pushed.

- **Jog feed and arrow keys**

The feed axis and direction corresponding to the arrow keys can be set with parameters (Nos. 7210 to 7215).

- **General purpose switches**

Eight optionally definable switches are added as an extended function of the software operator's panel. The name of these switches can be set by parameters as character strings of max. 8 characters. For the meanings of these switches, refer to the manual issued by machine tool builder.

11.5 SCREENS DISPLAYED BY FUNCTION KEY SYSTEM

When the CNC and machine are connected, parameters must be set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor or other parts.
This chapter describes how to set parameters on the MDI panel. Parameters can also be set with external input/output devices such as the Handy File (see Chapter 8).
In addition, pitch error compensation data used for improving the precision in positioning with the ball screw on the machine can be set or displayed by the operations under function key SYSTEM .

See Chapter 7 for the diagnostic screens displayed by pressing function key SYSTEM .

11.5.1 Displaying and Setting Parameters

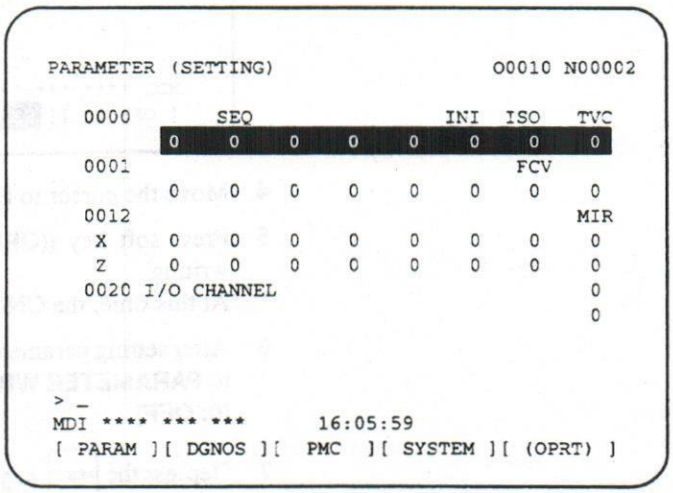
When the CNC and machine are connected, parameters are set to determine the specifications and functions of the machine in order to fully utilize the characteristics of the servo motor. The setting of parameters depends on the machine. Refer to the parameter list prepared by the machine tool builder.
Normally, the user need not change parameter setting.

Procedure for displaying and setting parameters

1 Set 1 for **PARAMETER WRITE** to enable writing. See the procedure for enabling/disabling parameter writing described below.

2 Press function key [SYSTEM] .

3 Press chapter selection soft key **[PARAM]** to display the parameter screen.


PARAMETER (SETTING) O0010 N00002
0000 SEQ INI ISO TVC
0 0 0 0 0 0 0 0
0001 FCV
0 0 0 0 0 0 0 0
0012 MIR
X 0 0 0 0 0 0 0 0
Z 0 0 0 0 0 0 0 0
0020 I/O CHANNEL 0
0
> _
MDI **** *** *** 16:05:59
[PARAM][DGNOS][PMC][SYSTEM][(OPRT)]

4 Move the cursor to the parameter number to be set or displayed in either of the following ways:
- Enter the parameter number and press soft key **[NO.SRH]** .
- Move the cursor to the parameter number using the page keys, [PAGE↑] and [PAGE↓] , and cursor keys, [↑], [↓], [←], and [→] .

5 To set the parameter, enter a new value with numeric keys and press soft key **[INPUT]** in the MDI mode. The parameter is set to the entered value and the value is displayed.

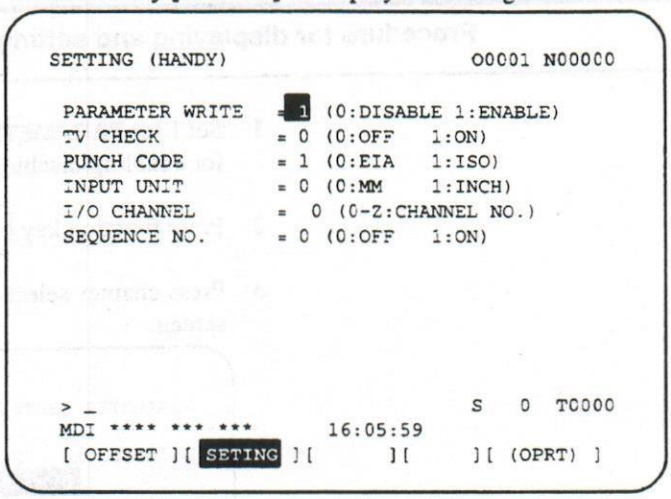
6 Set 0 for **PARAMETER WRITE** to disable writing.

Procedure for enabling/displaying parameter writing

1 Select the **MDI** mode or enter state emergency stop.

2 Press function key [OFFSET SETTING] .

3 Press soft key **[SETING]** to display the setting screen.


SETTING (HANDY) O0001 N00000
PARAMETER WRITE = 1 (0:DISABLE 1:ENABLE)
TV CHECK = 0 (0:OFF 1:ON)
PUNCH CODE = 1 (0:EIA 1:ISO)
INPUT UNIT = 0 (0:MM 1:INCH)
I/O CHANNEL = 0 (0-Z:CHANNEL NO.)
SEQUENCE NO. = 0 (0:OFF 1:ON)
> _
S 0 T0000
MDI **** *** ***
16:05:59
[OFFSET][SETING][ ][ ][(OPRT)]

4 Move the cursor to **PARAMETER WRITE** using cursor keys.

5 Press soft key **[(OPRT)]**, then press **[1: ON]** to enable parameter writing.
At this time, the CNC enters the P/S alarm state (No. 100).

6 After setting parameters, return to the setting screen. Move the cursor to **PARAMETER WRITE** and press soft key **[(OPRT)]** , then press **[0: OFF]**.

7 Depress the [RESET] key to release the alarm condition. If alarm No. 000 has occurred, however, turn off the power supply and then turn it on, otherwise the alarm is not released.

Explanations

- **Setting parameters with external input/output devices**

  See Chapter 8 for setting parameters with external input/output devices such as the Handy File.

- **Parameters that require turning off the power**

  Some parameters are not effective until the power is turned off and on again after they are set. Setting such parameters causes alarm 000. In this case, turn off the power, then turn it on again.

- **Parameter list**

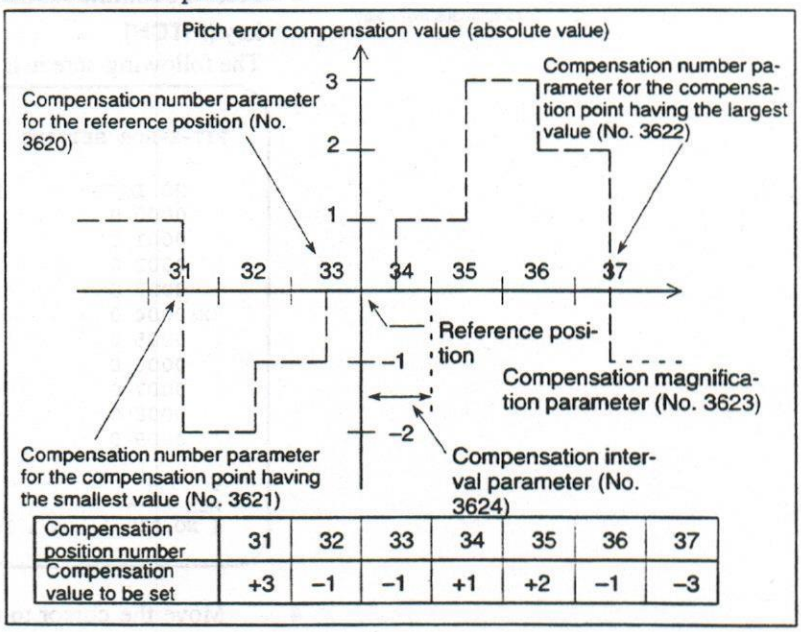
  Refer to the FANUC Series 20–TA Parameter Manual (B–62210E) for the parameter list.

- **Setting data**

  Some parameters can be set on the setting screen if the parameter list indicates "Setting entry is acceptable". Setting 1 for **PARAMETER WRITE** is not necessary when three parameters are set on the setting screen.

11.5.2 Displaying and Setting Pitch Error Compensation Data

If pitch error compensation data is specified, pitch errors of each axis can be compensated in detection unit per axis.
Pitch error compensation data is set for each compensation point at the intervals specified for each axis. The origin of compensation is the reference position to which the tool is returned.
The pitch error compensation data is set according to the characteristics of the machine connected to the NC. The content of this data varies according to the machine model. If it is changed, the machine accuracy is reduced.
In principle, the end user must not alter this data.
Pitch error compensation data can be set with external devices such as the Handy File (see Chapter 8). Compensation data can also be written directly with the MDI panel.
The following parameters must be set for pitch error compensation. Set the pitch error compensation value for each pitch error compensation point number set by these parameters.
In the following example, 33 is set for the pitch error compensation point at the reference position.



Compensation position number	31	32	33	34	35	36	37
Compensation value to be set	+3	–1	–1	+1	+2	–1	–3

· Number of the pitch error compensation point at the reference position (for each axis): Parameter 3620

· Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621

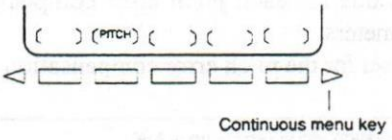
· Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622

·Pitch error compensation magnification (for each axis): Parameter 3623

· Interval of the pitch error compensation points (for each axis): Parameter 3624

Procedure for displaying and setting the pitch error compensation data

1 Set the following parameters:
- Number of the pitch error compensation point at the reference position (for each axis): Parameter 3620
- Number of the pitch error compensation point having the smallest value (for each axis): Parameter 3621
- Number of the pitch error compensation point having the largest value (for each axis): Parameter 3622
- Pitch error compensation magnification (for each axis): Parameter 3623
- Interval of the pitch error compensation points (for each axis): Parameter 3624



2 Press function key [SYSTEM].

3 Press the continuous menu key [▷], then press chapter selection soft key **[PITCH]**.
The following screen is displayed:

```
PIT-ERROR SETTING                    O0000 N00000

   NO. DATA        NO. DATA        NO. DATA
   0000 0          0010 0          0020 0
   0001 0          0011 0          0021 0
   0002 0          0012 0          0022 0
   0003 0          0013 0          0023 0
(X)0004 0          0014 0          0024 0
   0005 0          0015 0          0025 0
   0006 0          0016 0          0026 0
   0007 0          0017 0          0027 0
   0008 0          0018 0          0028 0
   0009 0          0019 0          0029 0

>_
MEM **** *** ***       16:05:59
[ NO.SRH ][ ON:1 ][ OFF:0 ][ +INPUT ][ -INPUT ]
```

4 Move the cursor to the compensation point number to be set in either of the following ways:
- Enter the compensation point number and press the [NO.SRH] soft key.
- Move the cursor to the compensation point number using the page keys, [PAGE↑] and [PAGE↓], and cursor keys, [↑], [↓], [←], and [→].

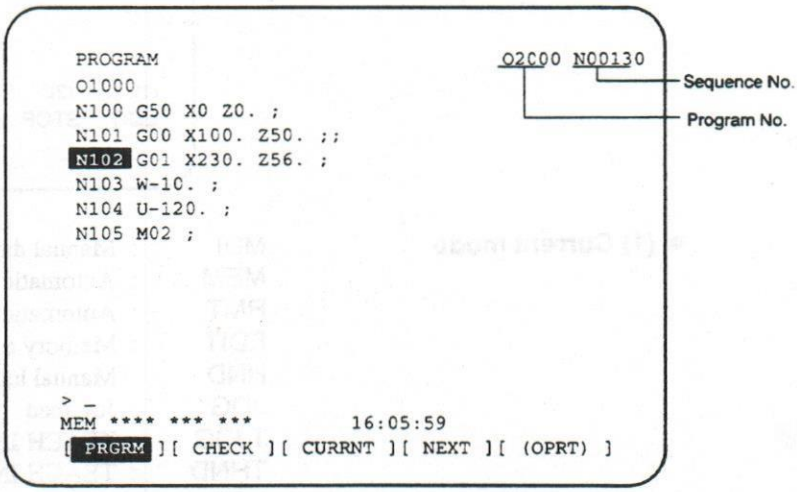
5 Enter a value with numeric keys and press the **[INPUT]** soft key.

11.6 DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION

The program number, sequence number, and current CNC status are always displayed on the screen except when the power is turned on, a system alarm occurs, or the PMC screen or the custom screen is displayed. If data setting or the input/output operation is incorrect, the CNC does not accept the operation and displays a warning message.
This section describes the display of the program number, sequence number, and status, and warning messages displayed for incorrect data setting or input/output operation.

11.6.1 Displaying the Program Number and Sequence Number

The program number and sequence number are displayed at the top right on the screen as shown below.



The program number and sequence number displayed depend on the screen and are given below:

On the program screen in the EDIT mode on Background edit screen :
The program No. being edited and the sequence number just prior to the cursor are indicated.

Other than above screens :
The program No. and the sequence No. executed last are indicated.

Immediately after program number search or sequence number search :

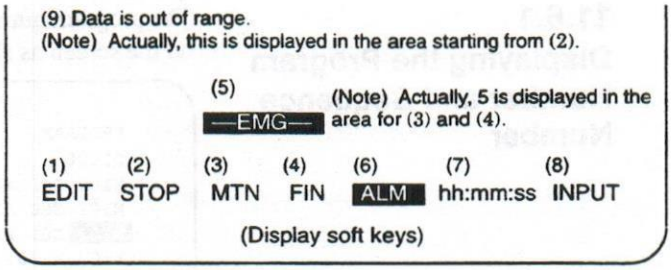
Immediately after the program No. search and sequence No. search, the program No. and the sequence No. searched are indicated.

11.6.2 Displaying the Status and Warning for Data Setting or Input/Output Operation

The current mode, automatic operation state, alarm state, and program editing state are displayed on the next to last line on the CRT screen allowing the operator to readily understand the operation condition of the system.
If data setting or the input/output operation is incorrect, the CNC does not accept the operation and a warning message is displayed on the next to last line of the CRT screen. This prevents invalid data setting and input/output errors.

Explanations

- **Description of each display**



- **(1) Current mode**

MDI	: Manual data input
MEM	: Automatic operation (MEMORY opration)
RMT	: Automatic operation (DNC operation)
EDIT	: Memory editing
HND	: Manual handle feed Manual incremental feed
JOG	: Jog feed
TJOG	: TEACH IN JOG
THND	: TEACH IN HANDLE
REF	: Manual reference position return

- **(2) Automatic operation status**

****	: Reset (When the power is turned on or the state in which program execution has terminated and automatic operation has terminated.)
STOP	: Automatic operation stop (The state in which one block has been executed and automatic operation is stopped.)
HOLD	: Feed hold (The state in which execution of one block has been interrupted and automatic operation is stopped.)
STRT	: Automatic operation start–up (The state in which the system operates automatically)

- **(3) Axis moving status/dwell status**

MTN	: Indicates that the axis is moving.
DWL	: Indicates the dwell state.
***	: Indicates a state other than the above.

- **(4) State in which an auxiliary function is being executed**

FIN	: Indicates the state in which an auxiliary function is being executed. (Waiting for the complete signal from the PMC)
***	: Indicates a state other than the above.

- **(5) Emergency stop or reset status**

—EMG—	: Indicates emergency stop.(Blinks in reversed display.)
—RESET—	: Indicates that the reset signal is being received.

- **(6) Alarm status**

  ALM : Indicates that an alarm is issued. (Blinks in reversed display.)
  BAT : Indicates that the battery is low. (Blinks in reversed display.)
  Space : Indicates a state other than the above.

- **(7) Current time**

  hh:mm:ss – Hours, minutes, and seconds

- **(8) Program editing status**

  INPUT : Indicates that data is being input.
  OUTPUT : Indicates that data is being output.
  SRCH : Indicates that a search is being performed.
  EDIT : Indicates that another editing operation is being performed (insertion, modification, etc.)
  LSK : Indicates that labels are skipped when data is input.
  Space : Indicates that no editing operation is being performed.

- **(9) Warning for data setting or input/output operation**

  When invalid data is entered (wrong format, value out of range, etc.), when input is disabled (wrong mode, write disabled, etc.), or when input/output operation is incorrect (wrong mode, etc.), a warning message is displayed. In this case, the CNC does not accept the setting or input/output operation.
  The following are examples of warning messages:

Example 1)
When a parameter is entered

```
> 1
EDIT    WRONG MODE
                    (Display soft keys)
```

Example 2)
When a parameter is entered

```
> 999999999
MDI    TOO MANY DIGITS
                    (Display soft keys)
```

Example 3)
When a parameter is output to an external input/output device

```
> _
MEM    WRONG MODE
                    (Display soft keys)
```

12 HELP FUNCTION

The help function displays on the screen detailed information about alarms issued in the CNC and about CNC operations. The following information is displayed.

- **Detailed information of alarms**

  When the CNC is operated incorrectly or an erroneous machining program is executed, the CNC enters the alarm state. The help screen displays detailed information about the alarm that has been issued and how to reset it. The detailed information is displayed only for a limited number of P/S alarms. These alarms are often misunderstood and are rather difficult to understand.

- **Operation method**

  If you are not sure about a CNC operation, refer to the help screen for information about each operation.

- **Parameter table**

  When setting or referring to a system parameter, if you are not sure of the number of the parameter, the help screen displays a list of parameter Nos. for each function.

Help Function Procedure

Procedure

1 Press the [HELP] key on the MDI panel. HELP (INITIAL MENU) screen is displayed.

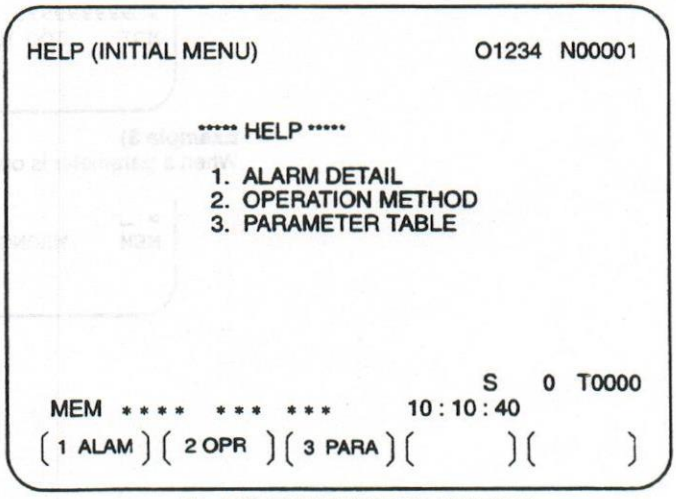

HELP (INITIAL MENU) O1234 N00001
***** HELP *****
1. ALARM DETAIL
2. OPERATION METHOD
3. PARAMETER TABLE
S 0 T0000
MEM **** *** *** 10:10:40
[1 ALAM] [2 OPR] [3 PARA] [ ] [ ]

Fig.12(a) HELP (INITIAL MENU) Screen

The user cannot switch the screen display from the PMC screen or CUSTOM screen to the help screen. The user can return to the normal CNC screen by pressing the [HELP] key or another function key.

ALARM DETAIL screen

2 Press soft key **[1 ALAM]** on the HELP (INITIAL MENU) screen to display detailed information about an alarm currently being raised.

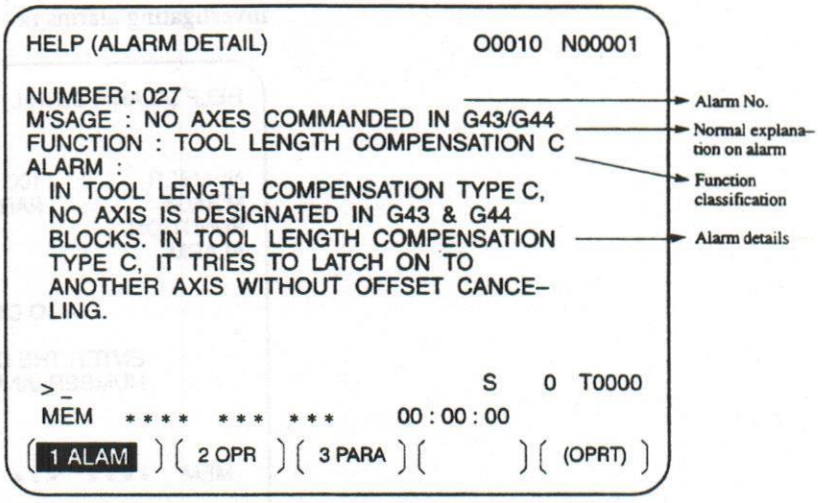


Fig.12(b) ALARM DETAIL Screen when Alarm No. 078 is issued

Note that only details of the alarm identified at the top of the screen are displayed on the screen.

If the alarms are all reset while the help screen is displayed, the alarm displayed on the ALARM DETAIL screen is deleted, indicating that no alarm is issued.

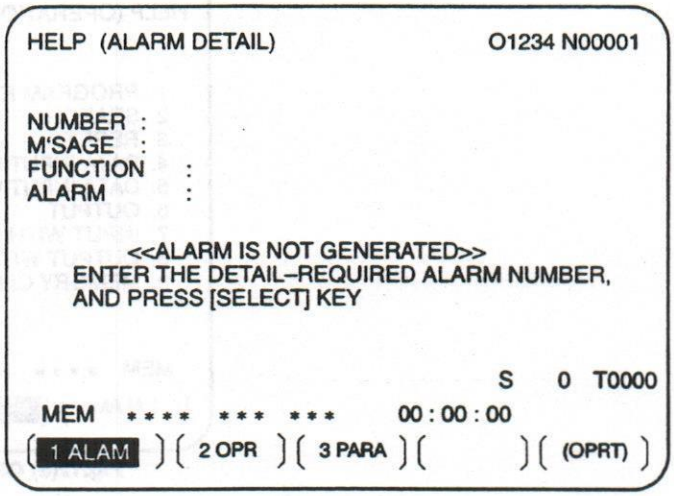


Fig.12(c) ALARM DETAIL Screen when No Alarm is issued

3 To get details on another alarm number, first enter the alarm number, then press soft key **[SELECT]**. This operation is useful for investigating alarms not currently being raised.

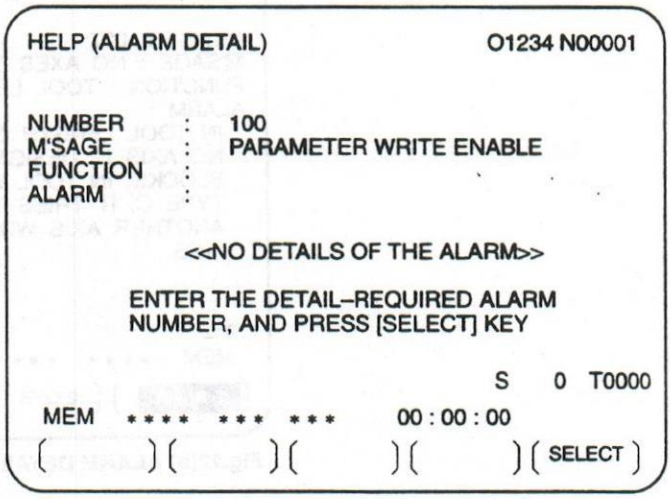

HELP (ALARM DETAIL) O1234 N00001
NUMBER : 100
M'SAGE : PARAMETER WRITE ENABLE
FUNCTION :
ALARM :
<<NO DETAILS OF THE ALARM>>
ENTER THE DETAIL-REQUIRED ALARM
NUMBER, AND PRESS [SELECT] KEY
S 0 T0000
MEM **** *** *** 00 : 00 : 00
( ) ( ) ( ) ( ) (SELECT)

Fig.12(d) ALARM DETAIL Screen when Alarm No.100 is selected

OPERATION METHOD screen

4 To determine an operating procedure for the CNC, press the soft key **[2 OPR]** key on the HELP (INITIAL MENU) screen. The OPERATION METHOD menu screen is then displayed. (See Fig. 13 (e).)

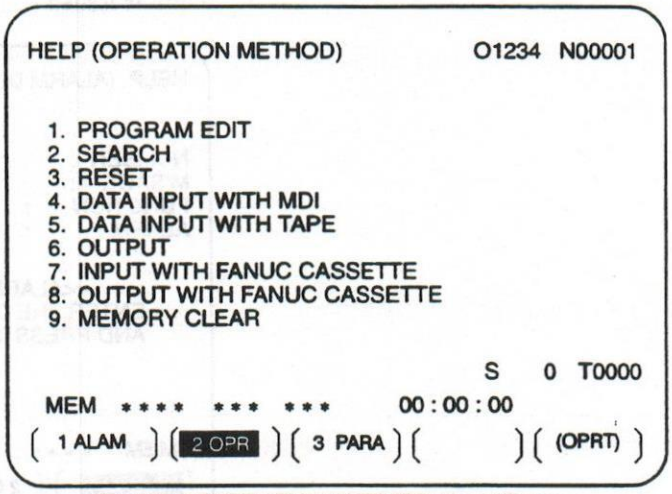

HELP (OPERATION METHOD) O1234 N00001
1. PROGRAM EDIT
2. SEARCH
3. RESET
4. DATA INPUT WITH MDI
5. DATA INPUT WITH TAPE
6. OUTPUT
7. INPUT WITH FANUC CASSETTE
8. OUTPUT WITH FANUC CASSETTE
9. MEMORY CLEAR
S 0 T0000
MEM **** *** *** 00 : 00 : 00
(1 ALAM) (2 OPR) (3 PARA) ( ) ((OPRT))

Fig.12(e) OPERATION METHOD Menu Screen

To select an operating procedure, enter an item No. from the keyboard then press the **[SELECT]** key.

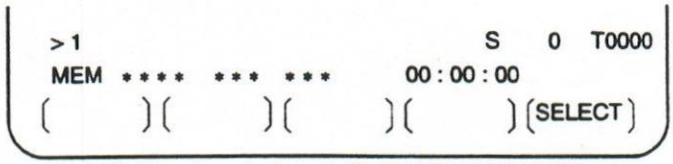


Fig.12(f) How to select each OPERATION METHOD screen

When "1. PROGRAM EDIT" is selected, for example, the screen in Figure 12 (g) is displayed.

On each OPERATION METHOD screen, it is possible to change the displayed page by pressing the PAGE key. The current page No. is shown at the upper right corner on the screen.

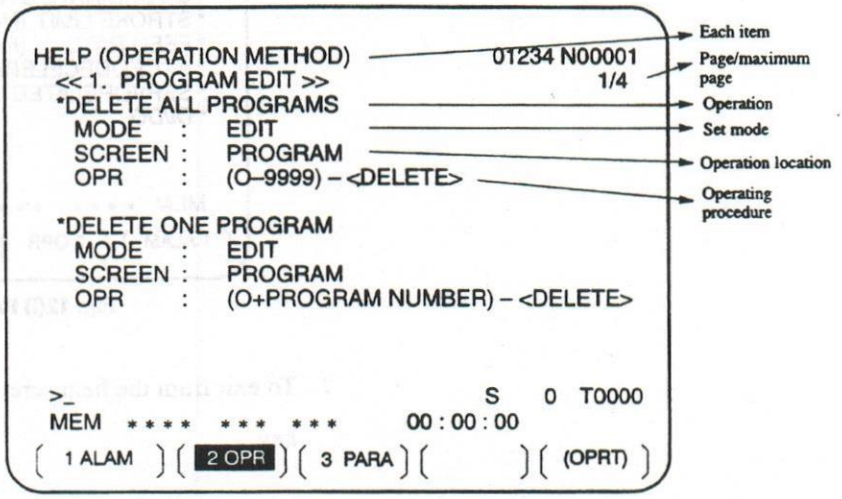
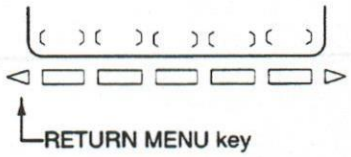


Fig.12(g) Selected OPERATION METHOD screen



5 To return to the OPERATION METHOD menu screen, press the RETURN MENU key to display **"[2 OPR]"** again, and then press the **[2 OPR]** key again.

To directly select another OPERATION METHOD screen on the screen shown in Figure 12 (g), enter an item No. from the keyboard and press the **[SELECT]** key.

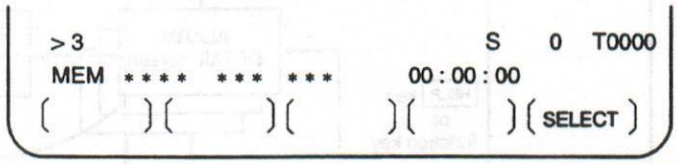


Fig.12(h) How to select another OPERATION METHOD screen

PARAMETER TABLE screen

6 If you are not sure of the No. of a system parameter to be set, or to refer to a system parameter, press the **[3 PARA]** key on the HELP (INITIAL MENU) screen. A list of parameter Nos. for each function is displayed. (See Figure 12 (i).)

It is possible to change the displayed page on the parameter screen. The current page No. is shown at the upper right corner on the screen.

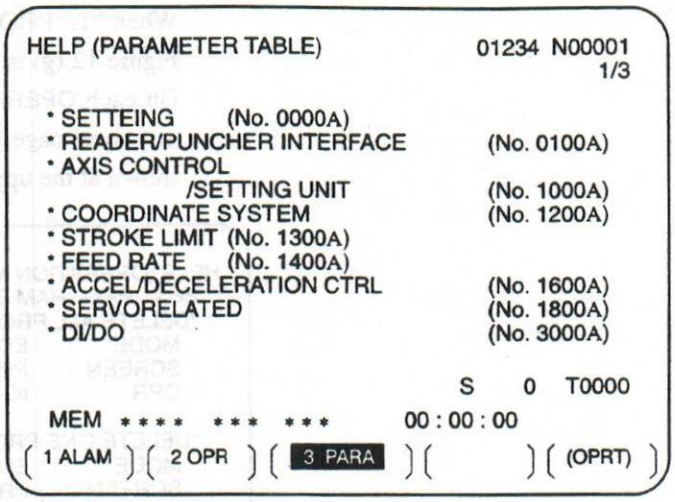
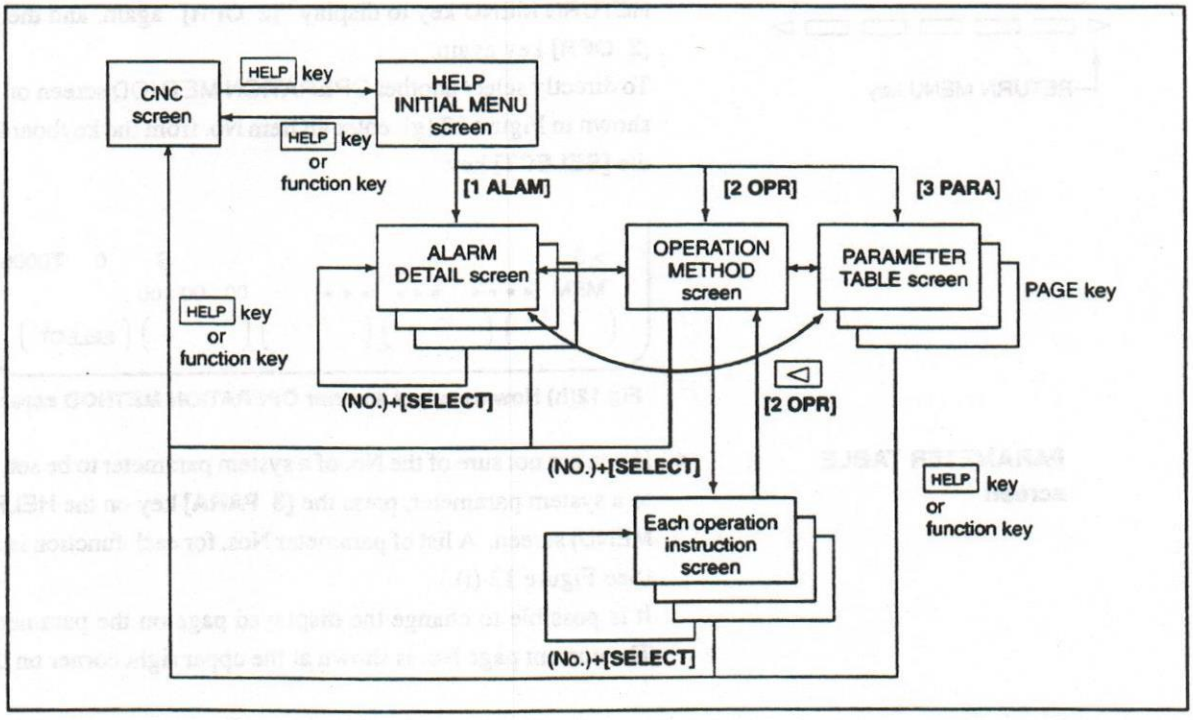


Fig. 12(i) PARAMETER TABLE screen

7 To exit from the help screen, press the HELP key or another function key.

Explanation

• Configuration of the Help Screen



IV. MAINTENANCE

1 METHOD OF REPLACING BATTERY

This chapter describes the method of replacing batteries as follows.

1.1 REPLACING CNC BATTERY FOR MEMORY BACK-UP
1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER
1.3 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER
(Servo amplifier converter unit only for Series 20 or α series Servo Amplifier Module)

- **Battery for Memory Backup (DC3V)**

Part programs, offset data, and system parameters are stored in CMOS memory in the control unit. The power to the CMOS memory is backed up by a lithium battery mounted on the front panel of the control unit. The above data is not lost even when the main battery goes dead. The backup battery is mounted on the control unit at shipping. This battery can maintain the contents of memory for about a year.

When the voltage of the battery becomes low, alarm message "BAL" blinks on the CRT display and the battery alarm signal is output to the PMC. When this alarm is displayed, replace the battery as soon as possible. In general, the battery can be replaced within two or three weeks, however, this depends on the system configuration.

If the voltage of the battery becomes any lower, memory can no longer be backed up and its contents are lost. Turning on the power to the control unit in this state causes system alarm 910 (SRAM parity alarm) to occur because the contents of memory are lost. Clear the entire memory and reenter data after replacing the battery.

- **Battery for Absolute pulse coder (DC6V)**

When the machine is equipped with absolute encoder system such as an absolute pulse coder or absolute linear scale, there is a battery for them separately from the battery for memory backup.
One battery unit can maintain absolute position data for six absolute pulse coders for a year.

When the voltage of the battery becomes low, APC alarms 3n6 to 3n8 (n: axis number) are displayed on the CRT display. When APC alarm 3n7 or 3n8 is displayed, replace the battery as soon as possible. In general, the battery should be replaced within two or three weeks, however, this depends on the number of pulse coders used.
If the voltage of the battery becomes any lower, the absolute positions for the pulse coders can no longer be maintained. Turning on the power to the control unit in this state causes APC alarm 3n0 (reference position return request alarm) to occur. Return the tool to the reference position after replacing the battery.

PO # 11518RRIP 5- NOV -2018

1.1 REPLACING CNC BATTERY FOR MEMORY BACK–UP

Replace CNC battery (lithium battery) for memory back–up by the following procedure.
Prepare lithium battery (A02B–0177–K106) in advance.
Observe the following precautions for lithium batteries:

Notes
If an unspecified battery is used, it may explode.
Replace the battery only with the specified battery (A02B–0177–K106).

Dispose of used lithium batteries as follows:
(1) Small quantities
Discharge the batteries and dispose of them as ordinary nonflammable garbage.
(2) Large quantities
Consult FANUC.

Procedure for replacing battery for memory back–up

1 Turn on machine (CNC) power.

Note
Replace the battery under the emergency stop state for safety, to escape the machine from moving during the replacement work.

2 Open the cover on the front panel of the power supply unit.

3 Remove the connector of CP8.

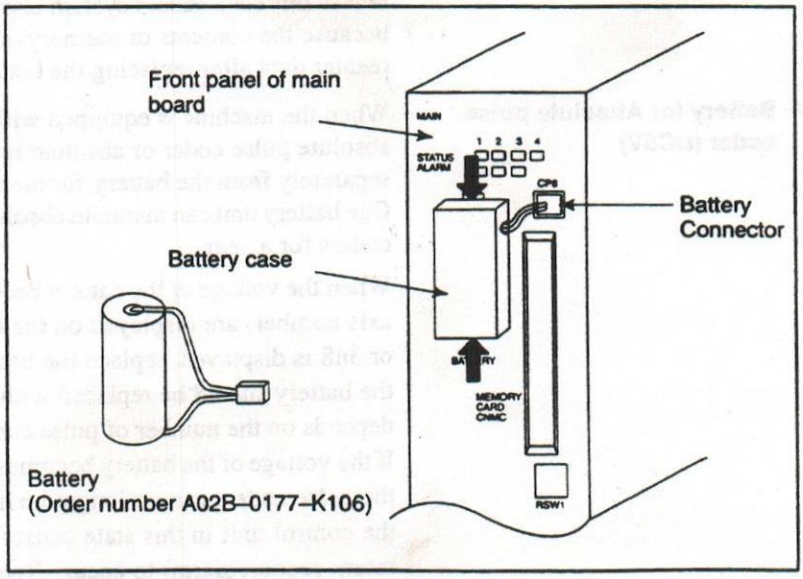


Fig. 1.1 (a) Replacing the battery (1)

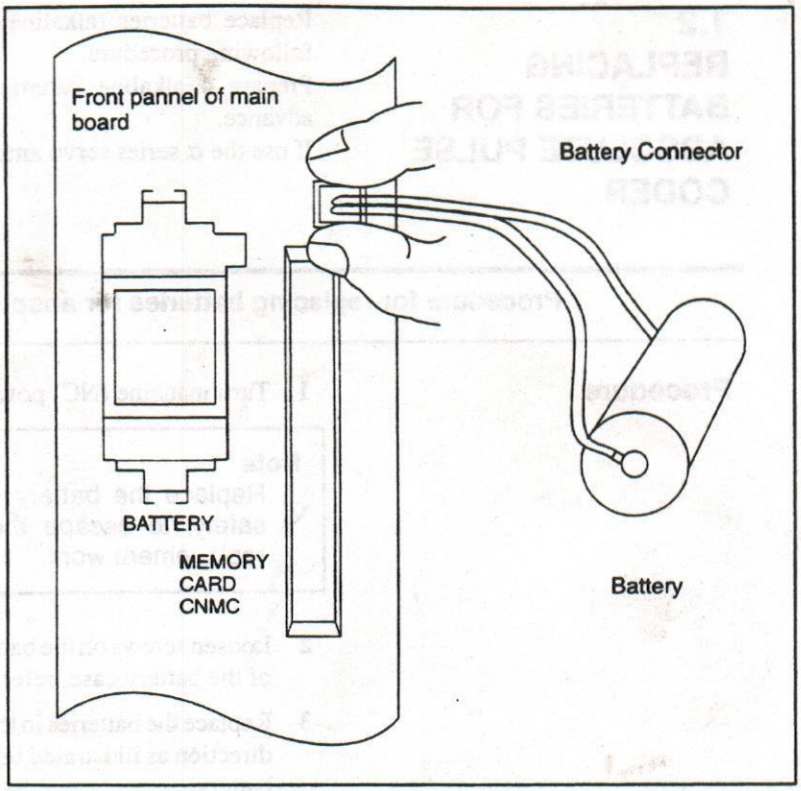


Fig. 1.1 (b) Replacing the battery (2)

4 Replace the battery and reconnect the connector.

5 Close the cover on the front panel of the power supply unit.

6 Turn off machine (CNC) power.

Notes

The batteries can be replaced whether NC power is ON or OFF. However, the replacement should be done within 30 minutes when the power is OFF. Memory contents might be erased if the NC power is OFF without battery for more than 30 minutes.

When the contents of memory are lost, turning on the power to the CNC may cause a RAM PARITY system alarm to occur. In this case, there is need to clear all memory contents and the power ON.

1.2 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER

Replace batteries (alkaline batteries) for absolute pulse coder by the following procedure.
Prepare 4 alkaline batteries (UM–1type) commercially available in advance.
If use the α series servo amplifier module, see section 1.3.

Procedure for replacing batteries for absolute pulse coder

Procedure

1 Turn machine (NC) power ON.

> **Note**
> Replace the battery under the emergency stop state for safety, to escape the machine from moving during the replacement work.

2 Loosen screws on the battery case to remove the cover. For placement of the battery case, refer to the machine tool builder's manual.

3 Replace the batteries in the case. Insert 2 batteries each in the opposite direction as illustrated below. Be sure to take notice of polarity of the battery.

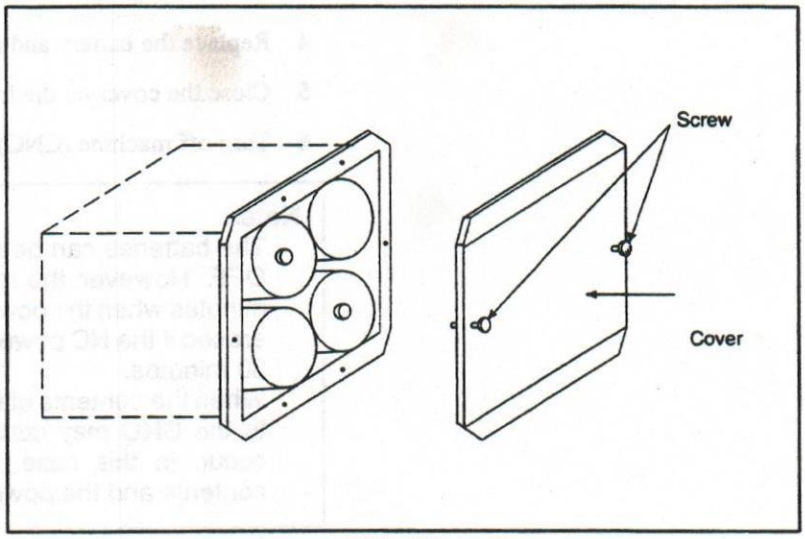


Fig. 1.2

4 After replacement, install the cover.

5 Turn off machine (NC) power OFF.

> **Notes**
> Replace the batteries for absolute pulse coder when NC power is ON.
> Replacing the batteries with power OFF causes the absolute position stored in memory to be lost.

1.3 REPLACING BATTERIES FOR ABSOLUTE PULSE CODER (SERVO AMPLIFIER CONVERTER UNIT ONLY FOR SERIES 20 OR α SERIES SERVO AMPLIFIER MODULE)

In case that the α series servo drive is used, the battery for absolute pulse coder could be provided on the servo amplifier converter unit only for Series 20 or α series servo amplifier module instead of the battery case as shown in 1.3. In this case the battery is not an alkaline battery but a lithium battery, A06B–6073–K001. Prepare the battery in advance and replace it by the following procedure.
Observe the following precautions for lithium batteries:

Notes
If an unspecified battery is used, it may explode.
Replace the battery only with the specified battery (A06B–6073–K001).

Dispose of used lithium batteries as follows:

(1) Small quantities
Discharge the batteries and dispose of them as ordinary nonflammable garbage.

(2) Large quantities
Consult FANUC.

Procedure for replacing batteries for absolute pulse coder

Procedure

1 Turn on machine (CNC) power.

Note
Replace the battery under the emergency stop state for safety, to escape the machine from moving during the replacement work.

2 Remove the battery case on the front panel of servo amplifier converter unit only for Series 20 or α series servo amplifier module (SVM).
The battery case can be removed by holding the top of the case and pulling the case towards you.

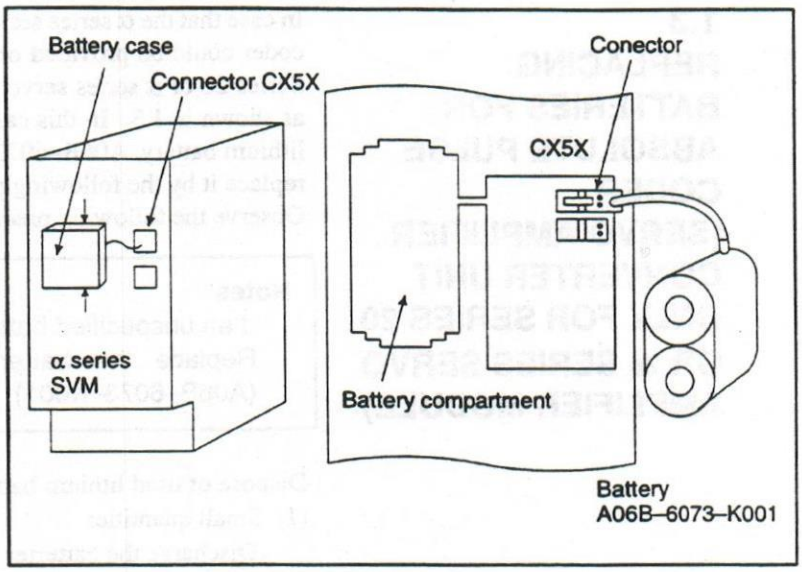


Fig. 1.3

3 Remove the connector of CX5X.

4 Replace the battery, and connect the connector.

5 Attach the battery case.

6 Turn off machine (CNC) power.

Notes

1. **Replace the batteries for absolute pulse coder when NC power is ON.**
   Replacing the batteries with power OFF causes the absolute position stored in memory to be lost.
2. **If your machine is equipped with a separate battery case follow the instructions in 1.2.**

APPENDIX

A TAPE CODE LIST

ISO code										EIA code									Meaning		
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1		Without custom macro B	With custom macro B
0			○	○		○				0			○			○				Number 0		
1	○		○	○		○			○	1						○			○	Number 1		
2	○		○	○		○		○		2						○		○		Number 2		
3			○	○		○		○	○	3				○		○		○	○	Number 3		
4	○		○	○		○	○			4						○	○			Number 4		
5			○	○		○	○		○	5				○		○	○		○	Number 5		
6			○	○		○	○	○		6				○		○	○	○		Number 6		
7	○		○	○		○	○	○	○	7						○	○	○	○	Number 7		
8	○		○	○	○	○				8					○	○				Number 8		
9			○	○	○	○			○	9				○	○	○			○	Number 9		
A		○				○			○	a		○	○			○			○	Address A		
B		○				○		○		b		○	○			○		○		Address B		
C	○	○				○		○	○	c		○	○	○		○		○	○	Address C		
D		○				○	○			d		○	○			○	○			Address D		
E	○	○				○	○		○	e		○	○	○		○	○		○	Address E		
F	○	○				○	○	○		f		○	○	○		○	○	○		Address F		
G		○				○	○	○	○	g		○	○			○	○	○	○	Address G		
H		○			○	○				h		○	○		○	○				Address H		
I	○	○			○	○			○	i		○	○	○	○	○			○	Address I		
J	○	○			○	○		○		j		○		○		○		○	○	Address J		
K		○			○	○		○	○	k		○		○		○		○		Address K		
L	○	○			○	○	○			l		○				○		○	○	Address L		
M		○			○	○	○		○	m		○		○		○	○			Address M		
N		○			○	○	○	○		n		○				○	○		○	Address N		
O	○	○			○	○	○	○	○	o		○				○	○	○		Address O		
P		○		○		○				p		○		○		○	○	○	○	Address P		
Q	○	○		○		○			○	q		○		○	○	○				Address Q		
R	○	○		○		○		○		r		○			○	○			○	Address R		
S		○		○		○		○	○	s				○	○			○		Address S		
T	○	○		○		○	○			t				○		○		○	○	Address T		
U		○		○		○	○		○	u				○	○	○	○			Address U		
V		○		○		○	○	○		v				○		○	○		○	Address V		
W	○	○		○		○	○	○	○	w				○		○	○	○		Address W		
X	○	○		○	○	○				x				○	○		○	○	○	Address X		
Y		○		○	○	○			○	y				○	○	○	○			Address Y		
Z		○		○	○	○		○		z				○		○	○		○	Address Z		

ISO code										EIA code										Meaning	
Character	8	7	6	5	4		3	2	1	Character	8	7	6	5	4		3	2	1	Without custom macro B	With custom macro B
DEL	○	○	○	○	○	○	○	○	○	Del		○	○	○	○	○	○	○	○	×	×
NUL						○				Blank						○				×	×
BS	○				○	○				BS			○		○	○		○		×	×
HT					○	○			○	Tab			○	○	○	○	○	○		×	×
LF or NL					○	○		○		CR or EOB	○					○					
CR	○				○	○	○		○	___										×	×
SP	○		○			○				SP				○		○				□	□
%	○		○			○	○		○	ER					○	○		○	○		
(			○		○	○				(2-4-5)				○	○	○		○			
)	○		○		○	○			○	(2-4-7)		○			○	○		○			
+			○		○	○		○	○	+		○	○	○		○				Δ	
–			○		○	○	○		○	–		○				○					
:			○	○	○	○		○		___											
/	○		○		○	○	○	○	○	/			○	○		○			○		
.			○		○	○	○	○		.		○	○		○	○		○	○		
#	○		○			○		○	○	Parameter (No.6012)											
$			○			○	○			___										Δ	○
&	○		○			○	○	○		&					○	○	○	○		Δ	○
∇			○			○	○	○	○	___										Δ	○
*	○		○		○	○		○		Parameter (No.6010)										Δ	
,	○		○		○	○	○			,			○	○	○	○		○	○		
;	○		○	○	○	○		○	○	___										Δ	Δ
<			○	○	○	○	○			___										Δ	Δ
=	○		○	○	○	○	○		○	Parameter (No.6011)										Δ	
>	○		○	○	○	○	○	○		___										Δ	Δ
?			○	○	○	○	○	○	○	___										Δ	○
@	○	○				○				___										Δ	○
"			○					○		___										Δ	Δ
"	○	○		○	○	○		○	○	Parameter (No.6013)										Δ	
"	○	○		○	○	○		○	○	Parameter (No.6014)										Δ	

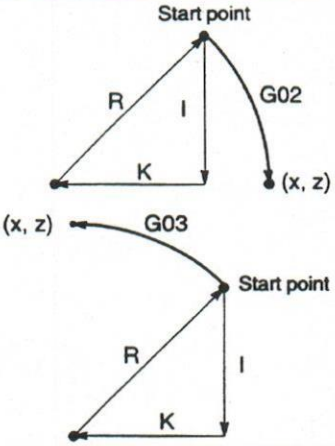
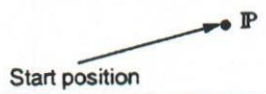
Notes

1. The symbols used in the remark column have the following meanings.
   (Space) : The character will be registered in memory and has a specific meaning. If it is used incorrectly in a statement other than a comment, an alarm occurs.
   × : The character will not be registered in memory, but will be ignored.
   Δ : The character will be registered in memory, but will be ignored during program execution.
   ○ : The character will be registered in memory. If it is used in a statement other than a comment, an alarm occurs.
   □ : If it is used in a statement other than a comment, the character will not be registered in memory. If it is used in a comment, it will be registered in memory.
2. Codes not in this table are ignored if their parity is correct.
3. Codes with incorrect parity cause the TH alarm. But they are ignored without generating the TH alarm when they are in the comment section.
4. A character with all eight holes punched is ignored and does not generate TH alarm in EIA code.

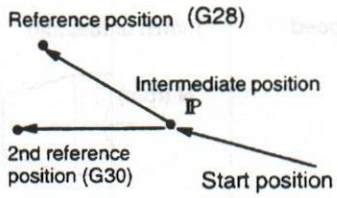
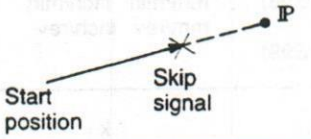
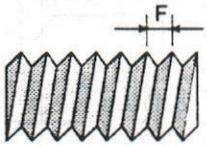
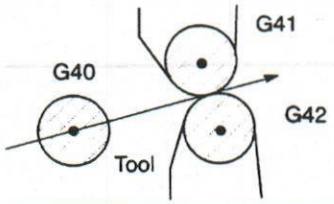
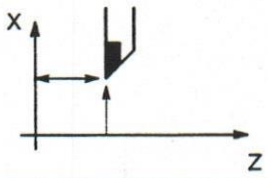
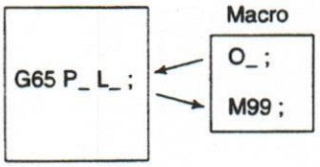
B LIST OF FUNCTIONS AND TAPE FORMAT

In the tables below, IP _ :presents a combination of arbitrary axis addresses using X and Z.
x = 1st basic axis (X usually)
z = 2nd basic axis (Z usually)

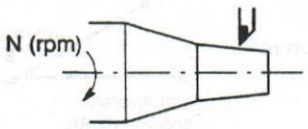
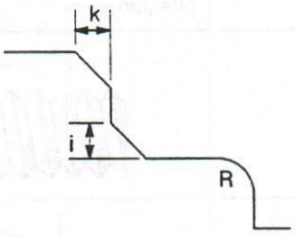
(1/3)

Functions	Illustration	Tape format
Positioning (G00)		G00 IP_ ;
Linear interpolation (G01)		G01 IP_ F_;
Circular interpolation (G02, G03)		{G02 / G03} X_ Z_ {R_ / I_ K_} F_ ;
Dwell (G04)		G04 {X_ / P_} ;
Change of offsetvalue by program(G10)		Tool geometry offset value G10 P_ X_ Z_ R_ Q_ ; P=1000+Geometry offset number Tool wear offset value G10 P_ X_ Z_ R_ Q_ ; P=Wear offset number
Inch/metric conversion (G20, G21)		Inch input : G20 Metric input : G21
Reference position return check (G27)		G27 IP_ ;

(2/3)

Functions	Illustration	Tape format
Reference position return (G28) 2nd, reference position return (G30)		G28 IP_ ; G30 IP_ ;
Skip fubction (G31)		G31 IP_ F_;
Thread cutting (G32)		Equal lead thread cutting G32 IP_ F_;
Cutter compensation (G40, G41, G42)		{G41 / G42} P_ ; P : Tool offset number G40 : Cancel
Coordinate system setting Spindle speed setting (G50)		G50 X_ Z_ ; Coordinate system setting G50 S_ ; Spindle speed setting
Custom macro (G65, G66, G67)		One–shot call G65 P_ L_ < Argument assignment>; P: Program No. L: Number of repetition Model call {G66 / G66.1} P_ L_ <Argument assignment> ; G67 ; ... Cancel
Canned cycle (G90, G92, G94)	Refer to II.14. FUNCTIONS TO SIMPLIFY PROGRAMMING	{G90 / G92 / G94} X_ Z_ R_ F_ ;

(3/3)

Functions	Illustration	Tape format
Constant surface speed control (G96/G97)	m/min or feet/min 	G96 S_ ; G97 ; Cancel
Feed per minute (G98) Feed per revolution (G99)	mm/min inch/min mm/rev inch/rev	G98 ... F_ ; G99 ... F_ ;
Chamfering, Corner R		X_ { C(K) ± k / R_ } ; Z_ { C(I) ± i / R_ } ;

C RANGE OF COMMAND VALUE

Linear axis

- **In case of millimeter input, feed screw is millimeter**

	Increment system
	IS-B
Least input increment	0.001 mm
Least command increment	X : 0.0005 mm Y : 0.001 mm
Max. programmable dimension	±99999.999 mm
Max. rapid traverse *1	240000 mm/min
Feedrate range *1	Feed per minute : 1 to 240000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev
Incremental feed	0.001, 0.01, 0.1, 1mm/step
Tool compensation	0 to ±999.999 mm
Dwell time	0 to 99999.999 sec

- **In case of inch input, feed screw is millimeter**

	Increment system
	IS-B
Least input increment	0.0001 inch
Least command increment	X : 0.0005 inch Y : 0.001 inch
Max. programmable dimension	±9999.9999 inch
Max. rapid traverse *1	240000 mm/min
Feedrate range *1	Feed per minute : 0.01 to 9600 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step
Tool compensation	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec

- **In case of inch input, feed screw is inch**

	Increment system
	IS-B
Least input increment	0.0001 inch
Least command increment	X : 0.00005 inch Y : 0.0001 inch
Max. programmable dimension	±9999.9999 inch
Max. rapid traverse *1	9600 inch/min
Feedrate range *1	Feed per minute : 0.01 to 9600 inch/min Feed per revolution 0.000001 to 9.999999 inch/rev
Incremental feed	0.0001, 0.001, 0.01, 0.1 inch/step
Tool compensation	0 to ±99.9999 inch
Dwell time	0 to 99999.999 sec

- **In case of millimeter input, feed screw is inch**

	Increment system
	IS-B
Least input increment	0.001 mm
Least command increment	X : 0.00005 inch Y : 0.0001 inch
Max. programmable dimension	±99999.999 mm
Max. rapid traverse *1	9600 inch/min
Feedrate range *1	Feed per minute : 1 to 240000 mm/min Feed per revolution 0.0001 to 500.0000 mm/rev
Incremental feed	0.001, 0.01, 0.1, 1mm/step
Tool compensation	0 to ±999.999 mm
Dwell time	0 to 99999.999 sec

Note

Note

*1 The feedrate range shown above are limitations depending on CNC interpolation capacity.
As a whole system, limitations depending on servo system must also be considered.

D NOMOGRAPHS

D.1 INCORRECT THREADED LENGTH

The leads of a thread are generally incorrect in δ_1 and δ_2, as shown in Fig. D.1 (a), due to automatic acceleration and deceleration.
Thus distance allowances must be made to the extent of δ_1 and δ_2 in the program.

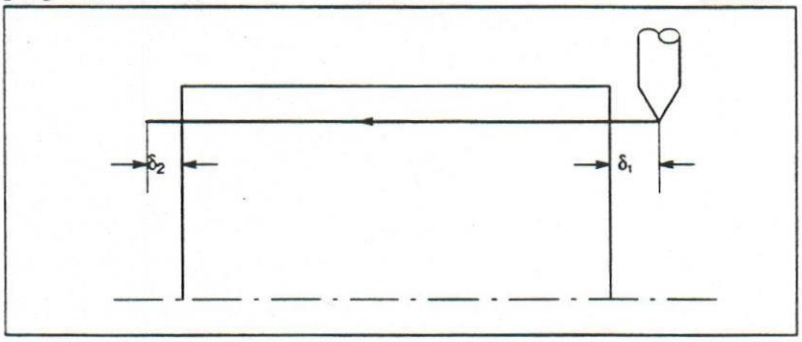


Fig.D.1(a) Incorrect thread position

Explanations

- **How to determine δ_2**

$$\delta_2 = T_1 V \text{ (mm)} \quad \cdots\cdots \quad (1)$$

$$V = \frac{1}{60} RL$$

T_1 : Time constant of servo system (sec)
V : Cutting speed (mm/sec)
R : Spindle speed (rpm)
L : Thread feed (mm)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

- **How to determine δ_1**

$$\delta_1 = \{t - T_1 + T_1 \exp(-\frac{t}{T_1})\} V \quad \cdots\cdots \quad (2)$$

$$a = \exp(-\frac{t}{T_1}) \quad \cdots\cdots \quad (3)$$

T_1 : Time constant of servo system (sec)
V : Cutting speed (mm/sec)

Time constant T_1 (sec) of the servo system: Usually 0.033 s.

The lead at the beginning of thread cutting is shorter than the specified lead L, and the allowable lead error is ΔL. Then as follows.

$$a = \frac{\Delta L}{L}$$

When the value of H*a*I is determined, the time lapse until the thread accuracy is attained. The time HtI is substituted in (2) to determine δ_1: Constants V and T_1 are determined in the same way as for δ_2. Since the calculation of δ_1 is rather complex, a nomography is provided on the following pages.

- **How to use nomograph**

First specify the class and the lead of a thread. The thread accuracy, *a*, will be obtained at (1), and depending on the time constant of cutting feed acceleration/ deceleration, the δ_1 value when V = 10mm / s will be obtained at (2). Then, depending on the speed of thread cutting, δ_1 for speed other than 10mm/ s can be obtained at (3).

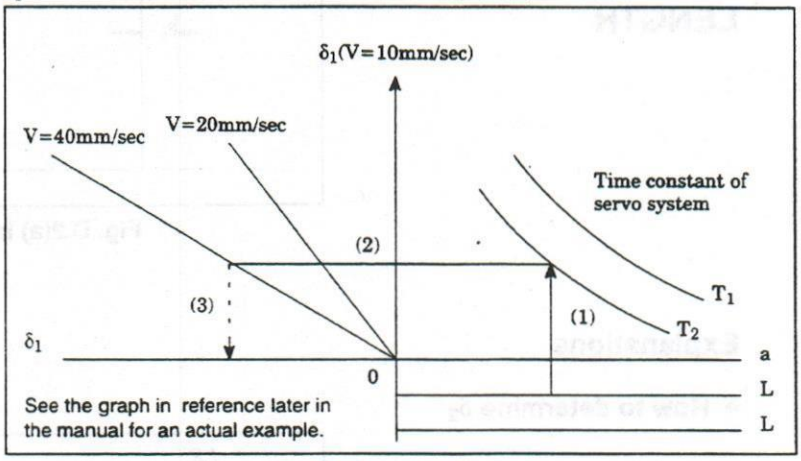


Fig.D.1(b) Nomograph

Notes
The equations for δ_1, and δ_2 are for when the acceleration/deceleration time constant for cutting feed is 0.

D.2 SIMPLE CALCULATION OF INCORRECT THREAD LENGTH

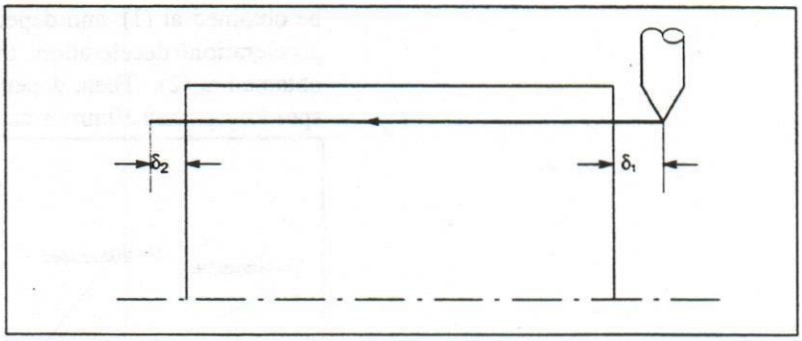


Fig. D.2(a) Incorrect threaded portion

Explanations

- **How to determine δ_2**

$$\delta_2 = \frac{LR}{1800\ *} \text{ (mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

- **How to determine δ_1**

$$\delta_1 = \frac{LR}{1800\ *}(-1-lna) \text{ (mm)}$$
$$= \delta_2(-1-lna) \text{ (mm)}$$

R : Spindle speed (rpm)
L : Thread lead (mm)

* When time constant T of the servo system is 0.033 s.

Following a is a permitted value of thread.

a	−1−lna
0.005	4.298
0.01	3.605
0.015	3.200
0.02	2.912

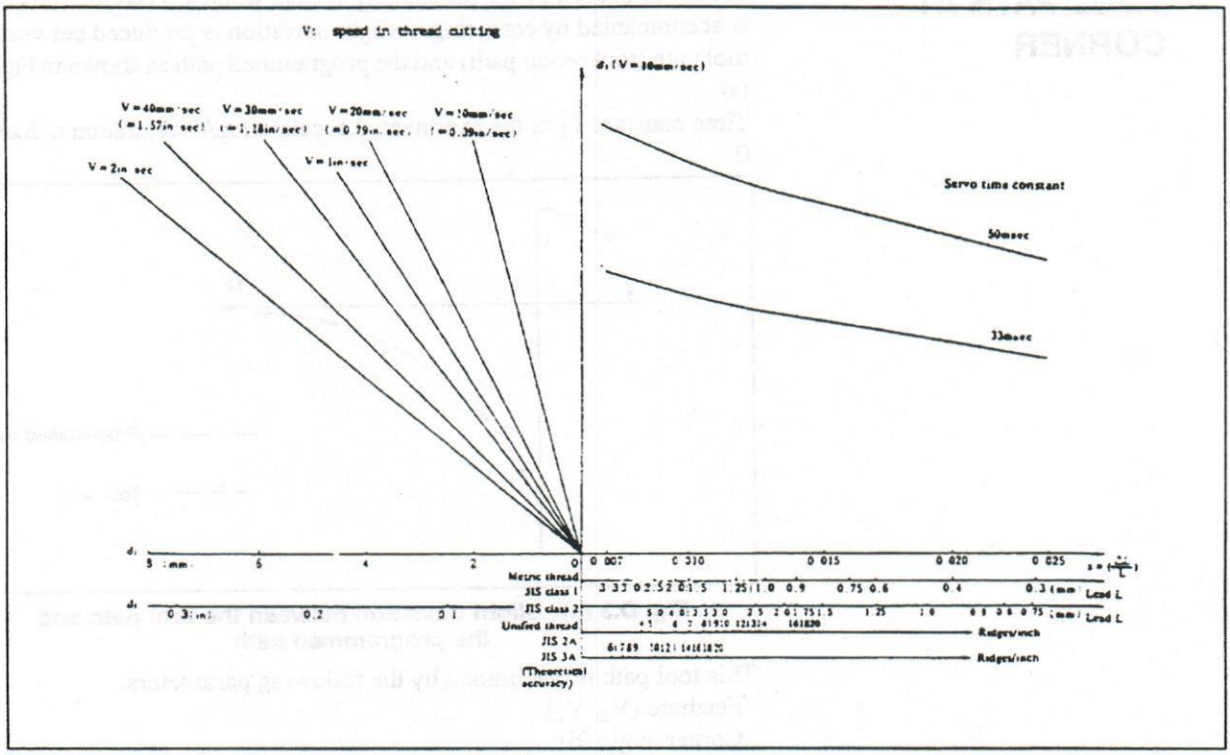
Examples

R=350rpm
L=1mm
a=0.01 then

$$\delta_2 = \frac{350 \times 1}{1800} = 0.194 \text{ (mm)}$$

$$\delta_1 = \delta_2 \times 3.605 = 0.701 \text{ (mm)}$$

- **Reference**



Nomograph for obtaining approach distance δ_1

D.3 TOOL PATH AT CORNER

When servo system delay (by exponential acceleration/deceleration at cutting or caused by the positioning system when a servo motor is used) is accompanied by cornering, a slight deviation is produced between the tool path (tool center path) and the programmed path as shown in Fig. D.3 (a).

Time constant T_1 of the exponential acceleration/deceleration is fixed to 0.

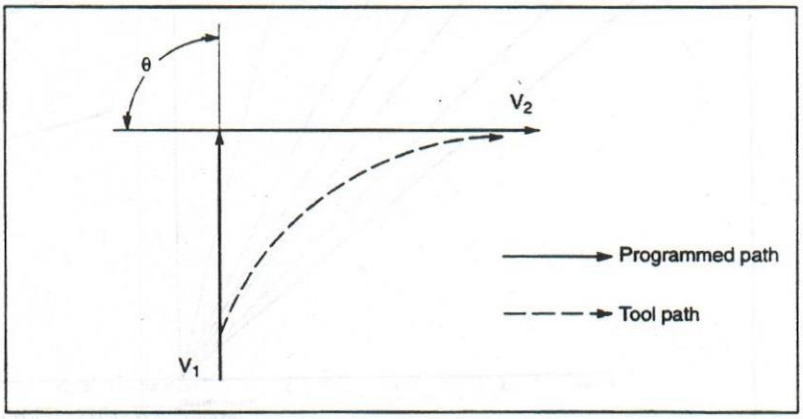


Fig. D.3 (a) Slight deviation between the tool path and the programmed path

This tool path is determined by the following parameters:

- Feedrate (V_1, V_2)
- Corner angle (θ)
- Exponential acceleration / deceleration time constant (T_1) at cutting ($T_1 = 0$)
- Presence or absence of buffer register.

The above parameters are used to theoretically analyze the tool path and above tool path is drawn with the parameter which is set as an example. When actually programming, the above items must be considered and programming must be performed carefully so that the shape of the workpiece is within the desired precision.

In other words, when the shape of the workpiece is not within the theoretical precision, the commands of the next block must not be read until the specified feedrate becomes zero. The dwell function is then used to stop the machine for the appropriate period.

Analysis

The tool path shown in Fig. D.3 (b) is analyzed based on the following conditions:
Feedrate is constant at both blocks before and after cornering.
The controller has a buffer register. (The error differs with the reading speed of the tape reader, number of characters of the next block, etc.)

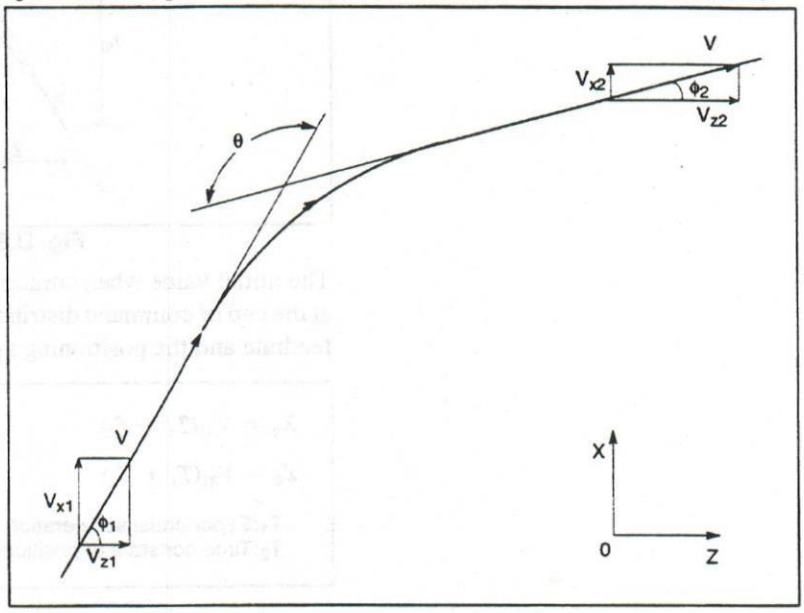


Fig. D.3(b) Example of tool path

- **Description of conditions and symbols**

$$V_{X1} = V\sin\phi_1$$

$$V_{Y1} = V\cos\phi_1$$

$$V_{X2} = V\sin\phi_2$$

$$V_{Y2} = V\cos\phi_2$$

V : Feedrate at both blocks before and after cornering
V_{X1} : X–axis component of feedrate of preceding block
V_{Z1} : Z–axis component of feedrate of preceding block
V_{X2} : X–axis component of feedrate of following block
V_{Z2} : Z–axis component of feedrate of following block
θ : Corner angle
ϕ_1 : Angle formed by specified path direction of preceding block and Z–axis
ϕ_2 : Angle formed by specified path direction of following block and Z–axis

- **Initial value calculation**

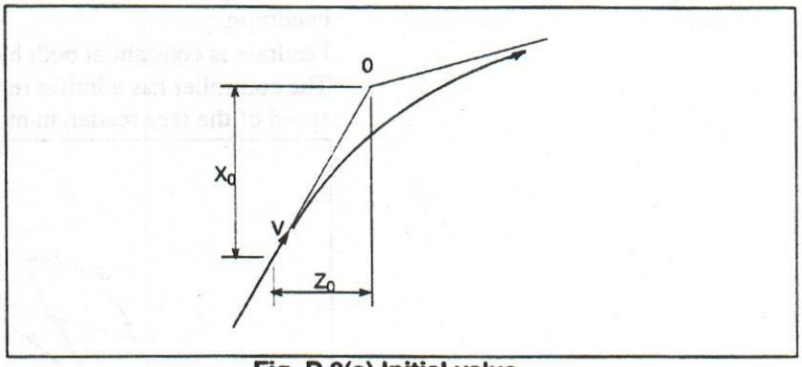


Fig. D.3(c) Initial value

The initial value when cornering begins, that is, the X and Z coordinates at the end of command distribution by the controller, is determined by the feedrate and the positioning system time constant of the servo motor.

$$X_0 = V_{X1}(T_1 + T_2)$$

$$Z_0 = V_{Z1}(T_1 + T_2)$$

T_1:Exponential acceleration / deceleration time constant. (T=0)
T_2:Time constant of positioning system (Inverse of position loop gain)

- **Analysis of corner tool path**

The equations below represent the feedrate for the corner section in X–axis direction and Y–axis direction.

$$V_X(t) = \frac{V_{X1} - V_{X2}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{X2}$$

$$V_Z(t) = \frac{V_{Z1} - V_{Z2}}{T_1 - T_2}\{T_1 \exp(-\frac{t}{T_1}) - T_2 \exp(-\frac{t}{T_2})\} + V_{Z2}$$

Therefore, the coordinates of the tool path at time t are calculated from the following equations:

$$X(t) = \int_0^t V_X(t)dt - X_0$$

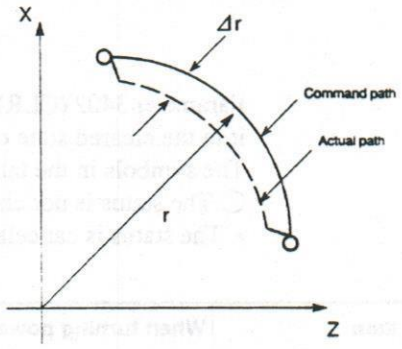
$$= \frac{V_{X2} - V_{X1}}{T_1 - T_2}\{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{X2}(T_1 + T_2 - t)$$

$$Z(t) = \int_0^t V_Z(t)dt - Z_0$$

$$= \frac{V_{Z2} - V_{Z1}}{T_1 - T_2}\{T_1^2 \exp(-\frac{t}{T_1}) - T_2^2 \exp(-\frac{t}{T_2})\} - V_{Z2}(T_1 + T_2 - t)$$

D.4 RADIUS DIRECTION ERROR AT CIRCLE CUTTING

When a servo motor is used, the positioning system causes an error between input commands and output results. Since the tool advances along the specified segment, an error is not produced in linear interpolation. In circular interpolation, however, radial errors may be produced, specially for circular cutting at high speeds.
This error can be obtained as follows:



$$\Delta r = \frac{1}{2}(T_1^2 + T_2^2)\frac{V^2}{r} \quad \cdots\cdots \quad (1)$$

Δr : Maximum radius error (mm)
v : Feedrate (mm/s)
r : Circle radius (mm)
T_1 : Exponential acceleration/deceleration time constant (sec) at cutting (T=0)
T_2 : Time constant of positoning system (sec). (Inverse of positon loop gain)

In the case of linear acceleration/deceleration after cutting feed interpolation, an approximation of this radius error can be obtained with the following expression:

$$\Delta r = (\frac{1}{24}T_1^2 + \frac{1}{2}T_2^2)\frac{V^2}{r}$$

Thus, the radius error in the case of linear acceleration/deceleration after interpolation is smaller than in
case of exponential acceleration/deceleration by a factor of 12, excluding any error caused by a servo loop time constant.

Since the machining radius r (mm) and allowable error Δr (mm) of the workpiece is given in actual machining, the allowable limit feedrate v (mm /sec) is determined by equation (1).
Since the acceleration/deceleration time constant at cutting which is set by this equipment varies with the machine tool, refer to the manual issued by the machine tool builder.

E STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET

Parameter 3402 (CLR) is used to select whether resetting the CNC places it in the cleared state or in the reset state (0: reset state/1: cleared state). The symbols in the tables below mean the following :
○:The status is not changed or the movement is continued.
×:The status is cancelled or the movement is interrupted.

Item		When turning power on	Cleared	Reset
Setting data	Offset value	○	○	○
	Data set by the MDI setting operation	○	○	○
	Parameter	○	○	○
Various data	Programs in memory	○	○	○
	Contents in the buffer storage	×	×	○ : MDI mode × : Other mode
	Display of sequence number	○	○ (Note 1)	○ (Note 1)
	One shot G code	×	×	×
	Modal G code	Initial G codes. (The G20 and G21 codes return to the same state they were in when the power was last turned off.)	Initial G codes. (G20/G21 are not changed.)	○
	F	Zero	Zero	○
	S, T, M	×	○	○
Work coordinate value		Zero	○	○
Action in operation	Movement	×	×	×
	Dwell	×	×	×
	Issuance of M, S and T codes	×	×	×
	Tool offset	×	Depending on parameter LVK(No.5003#6)	○ : MDI mode Other modes depend on parameter LVK(No.5003#6).
	Tool nose radius compensation	×	×	○ : MDI mode × : Other modes
	Storing called subprogram number	×	× (Note 2)	○ : MDI mode × : Other modes (Note 2)

Item		When turning power on	Cleared	Reset
Output signals	CNC alarm signal AL	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm	Extinguish if there is no cause for the alarm
	Reference position return completion LED	×	○ (× : Emergency stop)	○ (× : Emergency stop)
	S and T codes	×	○	○
	M code	×	×	×
	M, S and T strobe signals	×	×	×
	Spindle revolution signal (S analog signal)	×	○	○
	CNC ready signal MA	ON	○	○
	Servo ready signal SA	ON (When other than servo alarm)	ON (When other than servo alarm)	ON (When other than servo alarm)
	Cycle start LED (STL)	×	×	×
	Feed hold LED (SPL)	×	×	×

Notes

1. When heading is performed, the main program number is displayed.
2. When a reset is performed during execution of a subprogram, control returns the main program.
   Execution cannot be started from the middle of the subprogram.

F CHARACTER–TO–CODES CORRESPONDENCE TABLE

Character	Code	Comment	Character	Code	Comment
A	065		6	054	
B	066		7	055	
C	067		8	056	
D	068		9	057	
E	069			032	Space
F	070		!	033	Exclamation mark
G	071		"	034	Quotation mark
H	072		#	035	Hash sign
I	073		$	036	Dollar sign
J	074		%	037	Percent
K	075		&	038	Ampersand
L	076		'	039	Apostrophe
M	077		(	040	Left parenthesis
N	078		)	041	Right parenthesis
O	079		*	042	Asterisk
P	080		+	043	Plus sign
Q	081		,	044	Comma
R	082		–	045	Minus sign
S	083		.	046	Period
T	084		/	047	Slash
U	085		:	058	Colon
V	086		;	059	Semicolon
W	087		<	060	Left angle bracket
X	088		=	061	Equal sign
Y	089		>	062	Right angle bracket
Z	090		?	063	Question mark
0	048		@	064	HAtI mark
1	049		[	091	Left square bracket
2	050		^	092	
3	051		o	093	Yen sign
4	052		]	094	Right square bracket
5	053		_	095	Underscore

G ALARM LIST

1) Program errors (P/S alarm)

Number	Message	Contents
000	PLEASE TURN OFF POWER	A parameter which requires the power off was input, turn off power.
001	TH PARITY ALARM	TH alarm (A character with incorrect parity was input). Correct the tape.
002	TV PARITY ALARM	TV alarm (The number of characters in a block is odd). This alarm will be generated only when the TV check is effective.
003	TOO MANY DIGITS	Data exceeding the maximum allowable number of digits was input. (Refer to the item of max. programmable dimensions.)
004	ADDRESS NOT FOUND	A numeral or the sign " – " was input without an address at the beginning of a block. Modify the program .
005	NO DATA AFTER ADDRESS	The address was not followed by the appropriate data but was followed by another address or EOB code. Modify the program.
006	ILLEGAL USE OF NEGATIVE SIGN	Sign " – " input error (Sign " – " was input after an address with which it cannot be used. Or two or more " – " signs were input.) Modify the program.
007	ILLEGAL USE OF DECIMAL POINT	Decimal point " . " input error (A decimal point was input after an address with which it can not be used. Or two decimal points were input.) Modify the program.
009	ILLEGAL ADDRESS INPUT	Unusable character was input in significant area. Modify the program.
010	IMPROPER G–CODE	An unusable G code or G code corresponding to the function not provided is specified. Modify the program.
011	NO FEEDRATE COMMANDED	Feedrate was not commanded to a cutting feed or the feedrate was inadequate. Modify the program.
020	OVER TOLERANCE OF RADIUS	In circular interpolation (G02 or G03), difference of the distance between the start point and the center of an arc and that between the end point and the center of the arc exceeded the value specified in parameter No. 3410.
021	ILLEGAL PLANE AXIS COMMANDED	An axis other than on ZX plane was specified during circular interpolation. The alarm occurs when data set in parameter No.1022 is in valid.
023	ILLEGAL RADIUS COMMAND	In circular interpolation by radius designation, negative value was commanded for address R. Modify the program.
029	ILLEGAL OFFSET VALUE	The offset values specified by T code is too large. Modify the program.
030	ILLEGAL OFFSET NUMBER	The offset number in T function specified for tool offset is too large. Modify the program.
031	ILLEGAL P COMMAND IN G10	In setting an offset amount by G10, the offset number following address P was excessive or it was not specified. Modify the program.
032	ILLEGAL OFFSET VALUE IN G10	In setting an offset amount by G10 or in writing an offset amount by system variables, the offset amount was excessive.
033	NO SOLUTION AT CRC	A point of intersection cannot be determined for tool nose radius compensation. Modify the program.

Number	Message	Contents
034	NO CIRC ALLOWED IN ST–UP / EXT BLK	The start up or cancel was going to be performed in the G02 or G03 mode in tool nose radius compensation. Modify the program.
035	CAN NOT COMMANDED G31	Skip cutting (G31) was specified in tool nose radius compensation mode. Modify the program.
038	INTERFERENCE IN CIRCULAR BLOCK	Overcutting will occur in tool nose radius compensation because the arc start point or end point coincides with the arc center. Modify the program.
039	CHF/CNR NOT ALLOWED IN NRC	Chamfering or corner R was specified with a start–up, a cancel, or switching between G41 and G42 in tool nose radius compensation. The program may cause overcutting to occur in chamfering or corner R. Modify the program.
040	INTERFERENCE IN G90/G94 BLOCK	Overcutting will occur in tool nose radius compensation in canned cycle G90 or G94. Modify the program.
041	INTERFERENCE IN NRC	Overcutting will occur in tool nose radius compensation. Modify the program.
046	ILLEGAL REFERENCE RETURN COMMAND	Other than P2 is commanded for 2nd reference position return command.
050	CHF/CNR NOT ALLOWED IN THRD BLK	Chamfering or corner R is commanded in the thread cutting block. Modify the program.
051	MISSING MOVE AFTER CHF/CNR	Improper movement or the move distance was specified in the block next to the chamfering or corner R block. Modify the program.
052	CODE IS NOT G01 AFTER CHF/ CNR	The block next to the chamfering or corner R block is not G01. Modify the program.
053	TOO MANY ADDRESS COMMANDS	In the chamfering and corner R commands, two or more of I, K and R are specified.
054	NO TAPER ALLOWED AFTER CHF/ CNR	A block in which chamfering in the specified angle or the corner R was specified includes a taper command. Modify the program.
055	MISSING MOVE VALUE IN CHF/ CNR	In chamfering or corner R block, the move distance is less than chamfer or corner R amount.
056	NO END POINT & ANGLE IN CHF/ CNR	In the chamfering common, I(K) is commanded for the X(Z) axis. Modifuy the program.
059	PROGRAM NUMBER NOT FOUND	In an external program number search or external workpiece number search, a specified program number was not found. Otherwise, a program specified for searching is being edited in background processing. Check the program number and external signal. Or discontinue the background editing.
060	SEQUENCE NUMBER NOT FOUND	Commanded sequence number was not found in the sequence number search. Check the sequence number.
061	ADDRESS P/Q NOT FOUND IN G70–G73	Address P or Q is not specified in G70, G71, G72, or G73 command. Modify the program.
062	ILLEGAL COMMAND IN G71–G76	1 The depth of cut in G71 or G72 is zero or negative value. 2 The repetitive count in G73 is zero or negative value. 3 The negative value is specified to Δi or Δk is zero in G74 or G75. 4 A value other than zero is specified to address U or W, though Δi or Δk is zero in G74 or G75. 5 A negative value is specified to Δd, though the relief direction in G74 or G75 is determined. 6. Zero or a negative value is specified to the height of thread or depth of cut of first time in G76. 7 The specified minimum depth of cut in G76 is greater than the height of thread. 8 An unusable angle of tool tip is specified in G76. Modify the program.

Number	Message	Contents
063	SEQUENCE NUMBER NOT FOUND	The sequence number specified by address P in G70, G71, G72, or G73 command cannot be searched. Modify the program.
064	SHAPE PROGRAM NOT MONOTONOUSLY	A target shape which is not monotonous increase or decrease was specified in a repetitive canned cycle (G71 or G72).
065	ILLEGAL COMMAND IN G71–G73	1 G00 or G01 is not commanded at the block with the sequence number which is specified by address P in G71, G72, or G73 command. 2. Address Z(W) or X(U) was commanded in the block with a sequence number which is specified by address P in G71 or G72, respectively. Modify the program.
066	IMPROPER G–CODE IN G71–G73	An unallowable G code was commanded between two blocks specified by address P in G71, G72, or G73. Modify the program.
067	CAN NOT ERROR IN MDI MODE	G70, G71, G72, or G73 command with address P and Q was specified. Modify the program.
069	FORMAT ERROR IN G70–G73	The final move command in the blocks specified by P and Q of G70, G71, G72, or G73 ended with chamfering or corner R.
070	NO PROGRAM SPACE IN MEMORY	The memory area is insufficient. Delete any unnecessary programs, then retry.
071	DATA NOT FOUND	The address to be searched was not found. Or the program with specified program number was not found in program number search. Check the data.
072	TOO MANY PROGRAMS	The number of programs to be stored exceeded 63. Delete unnecessary programs and execute program registration again.
073	PROGRAM NUMBER ALREADY IN USE	The commanded program number has already been used. Change the program number or delete unnecessary programs and execute program registration again.
074	ILLEGAL PROGRAM NUMBER	The program number is other than 1 to 9999. Modify the program number.
075	PROTECT	An attempt was made to register a program whose number was protected.
076	ADDRESS P NOT DEFINED	Address P (program number) was not commanded in the block which includes an M98, G65, or G66 command. Modify the program.
077	SUB PROGRAM NESTING ERROR	The subprogram was called in five folds. Modify the program.
078	NUMBER NOT FOUND	A program number or a sequence number which was specified by address P in the block which includes an M98, M99, M65 or G66 was not found. The sequence number specified by a GOTO statement was not found. Otherwise, a called program is being edited in background processing. Correct the program, or discontinue the background editing.
079	PROGRAM VERIFY ERROR	In memory or program collation,a program in memory does not agree with that read from an external I/O device. Check both the programs in memory and those from the external device.
085	COMMUNICATION ERROR	When entering data in the memory by using Reader / Puncher interface, an overrun, parity or framing error was generated. The number of bits of input data or setting of baud rate or specification No. of I/O unit is incorrect.
086	DR SIGNAL OFF	When entering data in the memory by using Reader / Puncher interface, the ready signal (DR) of reader / puncher was turned off. Power supply of I/O unit is off or cable is not connected or a P.C.B. is defective.
087	BUFFER OVERFLOW	When entering data in the memory by using Reader / Puncher interface, though the read terminate command is specified, input is not interrupted after 10 characters read. I/O unit or P.C.B. is defective.

Number	Message	Contents
090	REFERENCE RETURN INCOMPLETE	The reference position return cannot be performed normally because the reference position return start point is too close to the reference position or the speed is too slow. Separate the start point far enough from the reference position; or specify a sufficiently fast speed for reference position return. Check the program contents.
092	AXES NOT ON THE REFERENCE POINT	The commanded axis by G27 (Reference position return check) did not return to the reference position.
100	PARAMETER WRITE ENABLE	On the PARAMETER(SETTING) screen, PWE(parameter writing enabled) is set to 1. Set it to 0, then reset the system.
101	PLEASE CLEAR MEMORY	The power turned off while rewriting the memory by program edit operation. If this alarm has occurred, press <RESET> while pressing <PROG>, and only the program being edited will be deleted. Register the deleted program.
109	P/S ALARM	A value other than 0 or 1 was specified after P in the G08 code, or no value was specified.
111	CALCULATED DATA OVERFLOW	The result of calculation is out of the allowable range (-10^{47} to -10^{-29}, 0, and 10^{-29} to 10^{47}).
112	DIVIDED BY ZERO	Division by zero was specified. (including tan 90°) Modify the program.
113	IMPROPER COMMAND	A function which cannot be used in custom macro is commanded. Modify the program.
114	FORMAT ERROR IN MACRO	There is an error in other formats than <Formula>. Modify the program.
115	ILLEGAL VARIABLE NUMBER	A value not defined as a variable number is designated in the custom macro or in high–speed cycle cutting. Modify the program.
116	WRITE PROTECTED VARIABLE	The left side of substitution statement is a variable whose substitution is inhibited. Modify the program.
118	PARENTHESIS NESTING ERROR	The nesting of bracket exceeds the upper limit (quintuple). Modify the program.
119	ILLEGAL ARGUMENT	The SQRT argument is negative, BCD argument is negative, or other values than 0 to 9 are present on each line of BIN argument. Modify the program.
122	DUPLICATE MACRO MODAL–CALL	The macro modal call is specified in double. Modify the program.
123	CAN NOT USE MACRO COMMAND IN DNC	Macro control command is used during DNC operation. Modify the program.
124	MISSING END STATEMENT	DO – END does not correspond to 1 : 1. Modify the program.
125	FORMAT ERROR IN MACRO	<Formula> format is erroneous. Modify the program.
126	ILLEGAL LOOP NUMBER	In DOn, $1 \leqq n \leqq 3$ is not established. Modify the program.
127	NC, MACRO STATEMENT IN SAME BLOCK	NC and custom macro commands coexist. Modify the program.
128	ILLEGAL MACRO SEQUENCE NUMBER	The sequence number specified in the branch command was not 0 to 9999. Or, it cannot be searched. Modify the program.
129	ILLEGAL ARGUMENT ADDRESS	An address which is not allowed in <Argument Designation > is used. Modify the program.
131	TOO MANY EXTERNAL ALARM MESSAGES	Five or more alarms have generated in external alarm message. Consult the PMC ladder diagram to find the cause.
132	ALARM NUMBER NOT FOUND	No alarm No. concerned exists in external alarm message clear. Check the PMC ladder diagram.
133	ILLEGAL DATA IN EXT. ALARM MSG	Small section data is erroneous in external alarm message or external operator message. Check the PMC ladder diagram.
199	MACRO WORD UNDEFINED	Undefined macro word was used. Modify the custom macro.

Number	Message	Contents
224	RETURN TO REFERENCE POINT	Not returned to reference point before cycle start.
231	ILLEGAL FORMAT IN G10 OR L50	Any of the following errors occurred in the specified format at the programmable–parameter input. 1 Address N or R was not entered. 2 A number not specified for a parameter was entered. 3 The axis number was too large. 4 An axis number was not specified in the axis–type parameter. 5 An axis number was specified in the parameter which is not an axis type. 6 An attempt was made to reset bit 4 of parameter 3202 (NE9) or change parameter 3210 (PSSWD) when they are protected by a password. Correct the program.
233	DEVICE BUSY	When an attempt was made to use a unit such as that connected via the RS–232–C interface, other users were using it.
240	BP/S ALARM	Background editing was performed during MDI operation.
245	T–CODE NOT ALOWED IN THIS BLOCK	One of the G codes, G50, G10, and G04, which cannot be specified in the same block as a T code, was specified with a T code.
5010	END OF RECORD	The end of record (%) was specified.

2) Background edit alarm

Number	Message	Contents
070 to 074 085 to 087	BP/S alarm	BP/S alarm occurs in the same number as the P/S alarm that occurs in ordinary program edit.
140	BP/S alarm	It was attempted to select or delete in the background a program being selected in the foreground. **(Note)** Use background editing correctly.

Note

Alarm in background edit is displayed in the key input line of the background edit screen instead of the ordinary alarm screen and is resettable by any of the MDI key operation.

3) Absolute pulse coder (APC) alarm

Number	Message	Contents
300	nth–axis origin return	Manual reference position return is required for the nth–axis (n=1 – 8).
301	APC alarm: nth–axis communication	nth–axis (n=1 – 2) APC communication error. Failure in data transmission Possible causes include a faulty APC, cable, or servo interface module.
302	APC alarm: nth–axis over time	nth–axis (n=1 – 2) APC overtime error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
303	APC alarm: nth–axis framing	nth–axis (n=1 – 2) APC framing error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
304	APC alarm: nth–axis parity	nth–axis (n=1 – 2) APC parity error. Failure in data transmission. Possible causes include a faulty APC, cable, or servo interface module.
305	APC alarm: nth–axis pulse error	nth–axis (n=1 – 2) APC pulse error alarm. APC alarm. APC or cable may be faulty.
306	APC alarm: nth–axis battery voltage 0	nth–axis (n=1 – 2) APC battery voltage has decreased to a low level so that the data cannot be held. APC alarm. Battery or cable may be faulty.
307	APC alarm: nth–axis battery low 1	nth–axis (n=1 – 2) axis APC battery voltage reaches a level where the battery must be renewed. APC alarm. Replace the battery.
308	APC alarm: nth–axis battery low 2	nth–axis (n=1 – 2) APC battery voltage has reached a level where the battery must be renewed (including when power is OFF). APC alarm .Replace battery.
309	APC alarm: nth–axis ZRN IMPOSSIBL	nth–axis (n=1 – 4) A reference position return (annot be mode because the serial α pulse coder does not rotate more than one turn.

4) Serial pulse coder (SPC) alarms

When either of the following alarms is issued, a possible cause is a faulty serial pulse coder or cable.

Number	Message	Contents
350	SPC ALARM: n AXIS PULSE CODER	The n axis (axis 1–2) pulse coder has a fault. Refer to diagnosis display No. 202 or 204 for details.
351	SPC ALARM: n AXIS COMMUNICATION	n axis (axis 1–2) serial pulse coder communication error (data transmission fault) Refer to diagnosis display No. 203 for details.

· The details of serial α pulse coder alarm No.350

The explanation of serial pulse coder alarm No. 350 (pulse coder alarm) appears in the diagnosis display (Nos. 202 and 204). If alarm No. 351 is output together with alarm No. 350, see the explanations of alarm No. 351.

How to determine the cause of an alarm is explained in the FANUC Series 20 Maintenance Manual (B–62175E) and the Servo Amplifier Maintenance Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
0202					RCA			SPH

#3 (RCA): Count error. The serial α pulse coder may be defective.

#0 (SPH): Abnormal software phase data. The serial α pulse coder or feedback cable may be defective.

	#7	#6	#5	#4	#3	#2	#1	#0
0204				LDA	PMS			

#4 (LDA): Abnormality of the serial α pulse coder LED. The serial α pulse coder may be defective.

#3 (PMS): Abnormal feedback pulse. The serial α pulse coder or feedback cable (between the pulse coder and servo amplifier module, or between the servo amplifier module and main board) may be defective.

· The details of serial α pulse coder alarm No.351

The explanation of serial pulse coder alarm No. 351 (communication alarm) appears in the diagnosis display (No. 203).

How to determine the cause of an alarm is explained in the FANUC Series 20 Maintenance Manual (B–62175E) and the Servo Amplifier Maintenance Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
0203	DTE	CRC	STB					

#7 (DTE): No response is returned from the serial α pulse coder. Possible causes are the malfunctioning of the serial pulse coder, breakage of the signal cable, or a +5 V drop in the voltage being supplied to the pulse coder.

#6 (CRC): Serial communication failure.

#5 (STB): One of the following may be defective: signal cable (between the pulse coder and servo amplifier module, or between the servo amplifier module or main board), serial α pulse coder, servo interface module, servo control module, or main board.

5) Servo alarms

Number	Message	Contents
400	SERVO ALARM: n–TH AXIS OVERLOAD	The n–th axis (axis 1–2) overload signal is on. Refer to diagnosis display No. 201 for details.
401	SERVO ALARM: n–TH AXIS VRDY OFF	The n–th axis (axis 1–2) servo amplifier READY signal (DRDY) went off.
404	SERVO ALARM: n–TH AXIS VRDY ON	Even though the n–th axis (axis 1–8) READY signal (MCON) went off, the servo amplifier READY signal (DRDY) is still on. Or, when the power was turned on, DRDY went on even though MCON was off. Check that the servo interface module and servo amp are connected.
405	SERVO ALARM: (ZERO POINT RETURN FAULT)	Position control system fault. Due to an NC or servo system fault in the reference position return, there is the possibility that reference position return could not be executed correctly. Try again from the manual reference position return.
410	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–2) stops is larger than the set value. Note) Limit value must be set to parameter No.1829 for each axis.
411	SERVO ALARM: n–TH AXIS – EXCESS ERROR	The position deviation value when the n–th axis (axis 1–2) moves is larger than the set value. Note) Limit value must be set to parameter No.1828 for each axis.
413	SERVO ALARM: n–th AXIS – LSI OVERFLOW	The contents of the error register for the n–th axis (axis 1–2) are beyond of the range of -2^{31} to 2^{31}. This error usually occurs as the result of an improperly set parameters.
414	SERVO ALARM: n–TH AXIS – DETECTION RELATED ERROR	N–th axis (axis 1–2) digital servo system fault. Refer to diagnosis display No. 200 and No.201 for details.
415	SERVO ALARM: n–TH AXIS – EXCESS SHIFT	A speed higher than 511875 units/s was attempted to be set in the n–th axis (axis 1–2). This error occurs as the result of improperly set CMR.
416	SERVO ALARM: n–TH AXIS – DISCONNECTION	Position detection system fault in the n–th axis (axis 1–2) pulse coder (disconnection alarm). Refer to diagnosis display No. 200 and No. 201 for details.
417	SERVO ALARM: n–TH AXIS – PARAMETER INCORRECT	This alarm occurs when the n–th axis (axis 1–2) is in one of the conditions listed below. (Digital servo system alarm) 1) The value set in Parameter No. 2020 (motor form) is out of the specified limit. 2) A proper value (111 or –111) is not set in parameter No.2022 (motor revolution direction). 3) Illegal data (a value below 0, etc.) was set in parameter No. 2023 (number of speed feedback pulses per motor revolution). 4) Illegal data (a value below 0, etc.) was set in parameter No. 2024 (number of position feedback pulses per motor revolution). 5) Parameters No. 2084 and No. 2085 (flexible field gear rate) have not been set. 6) A value outside the limit of {1 to the number of control axes} or a non–continuous value (Parameter 1023 (servo axis number) contains a value out of the range from 1 to the number of axes, or an isolated value was set in parameter No. 1023 (servo axisnumber).

· Details of servo alarm No. 400, No. 414 and No. 416

The explanations of serial pulse coder alarm Nos. 400, 414, and 416 (pulse coder alarms) appear in the diagnosis display (Nos. 201, 202, and 204).
How to determine the cause of an alarm is explained in the FANUC Series 20 Maintenance Manual (B–62175E) and the Servo Amplifier Maintenance Manual.

	#7	#6	#5	#4	#3	#2	#1	#0
0200	OVL	LV	OVC	HCA	HVA	DCA	FBA	OFA

	#7	#6	#5	#4	#3	#2	#1	#0
0201	ALD			EXP				

	#7	#6	#5	#4	#3	#2	#1	#0
0204	RAM	OFS	MCC			FSA		

(1) Servo alarm No. 400 (Overload)

Bit converted to "1"	Alarm details
OVL	Overheat (Power supply module unit)
OVL + EXP	Overheat (Servo amplifier module unit)
OVL + ALD	Overheat (Servo motor)

(2) Servo alarm No. 414 (Detection error)

Bit converted to "1"	Alarm details
OFA	Digital servo overflow
DCA	Regenerative discharge circuit alarm
HVA	High voltage alarm (Power supply module unit)
HCA	Abnormal current alarm (Power supply module unit)
HCA + EXP	Abnormal current alarm (Servo amplifier module unit)
OVC	Overcurrent alarm
LV	Low voltage alarm (Power supply module unit) (Power source unit)
LV + EXP	Low voltage alarm (Servo amplifier module unit)
LV + ALD	Low voltage alarm (Power supply module unit) (Main unit)
LV + EXP + ALD	Low voltage alarm (Servo amplifier module unit) (Main unit)
FSA	Fan stop alarm (Power supply module unit)
FSA + EXP	Fan stop alarm (Servo amplifier module unit)
MCC	Charge failure (Power supply module unit)
OFS	Abnormal current detected in digital servo module
RAM	RAM check error in servo control module

(3) Servo alarm No. 416 (Disconnection alarm)

Bit converted to "1"	Alarm details
FBA	Pulse coder disconnection (Software)
FBA + ALD	Built–in serial pulse coder disconnection
FBA + EXP + ALD	Separate position detector disconnection

6) Over travel alarms

Number	Message	Contents
500	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side stored stroke limit I. (Parameter No.1320 or 1326 **Notes**)
501	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side stored stroke limit I. (Parameter No.1321 or 1327 **Notes**)
506	OVER TRAVEL : +n	Exceeded the n–th axis (axis 1–8) + side hardware OT.
507	OVER TRAVEL : –n	Exceeded the n–th axis (axis 1–8) – side hardware OT.

7) Overheat alarms

Number	Message	Contents
700	OVERHEAT: CONTROL UNIT	Control unit overheat Check that the fan motor operates normally, and clean the air filter.
701	OVERHEAT: FAN MOTOR	The fan motor on the top of the cabinet for the control unit is overheated. Check the operation of the fan motor and replace the motor if necessary.

8) Serial spindle alarms

Number	Message	Contents
749	S–SPINDLE LSI ERROR	A communication error occurred for the serial spindle. The cause may be the disconnection of an cable or the interruption of the power to the spindle amplifier. (Note) Unlike alarm No. 750, this alarm occurs when a serial communication alarm is detected after the spindle amplifier is normally activated.
750	SPINDLE SERIAL LINK START FAULT	This alarm is generated when the spindle control unit is not ready for starting correctly when the power is turned on in the system with the serial spindle. The four reasons can be considered as follows: 1) An improperly connected cable, or the spindle control unit's power is OFF. 2) When the NC power was turned on under alarm conditions other than SU–01 or AL–24 which are shown on the LED display of the spindle control unit. In this case, turn the spindle amplifier power off once and perform startup again. 3) Other reasons (improper combination of hardware) This alarm does not occur after the system including the spindle control unit is activated. See diagnostic display No. 409 for details.
751	FIRST SPINDLE ALARM DETECTION (AL–XX)	This alarm indicates in the NC that an alarm is generated in the spindle unit of the system with the serial spindle. The alarm is displayed in form AL–XX (XX is a number). Refer to **(11) Alarms displayed on spindle servo unit** .The alarm number XX is the number indicated on the spindle amplifier. The CNC holds this number and displays on the screen.

- **The details of spindle alarm No.750**

The details of spindle alarm No. 750 are displayed in the diagnosis display (No. 409) as shown below.

	#7	#6	#5	#4	#3	#2	#1	#0
409					SPE	S2E	S1E	SHE

SPE **0** : In the spindle serial control, the serial spindle parameters fulfill the spindle unit startup conditions.
1 : In the spindle serial control, the serial spindle parameters do not fulfill the spindle unit startup conditions.

S2E Not used (0 is displayed)

S1E **0** : The first spindle is normal during the spindle serial control startup.
1 : The first spindle was detected to have a fault during the spindle axis serial control startup.

SHE **0** : The serial communications module in the CNC is normal.
1 : The serial communications module in the CNC was detected to have a fault.

9) System alarms

(These alarms cannot be reset with reset key.)

Number	Message	Contents
900	ROM PARITY	F–ROM parity error in a ROM file (control software), such as CNC, macro, or digital servo. The F–ROM module may be defective.
910	DRAM PARITY : (Low)	For an SRAM parity error, initialize the memory. If the error subsequently recurs, or in the case of a DRAM parity error, replace the RAM module. Subsequently, set the parameters and all other data again.
911	DRAM PARITY: (High)	
912	SRAM PARITY: (Low)	
913	SRAM PARITY : (High)	
920	SERVO ALARM (1/2 AXIS)	Servo alarm (1st or 2nd axis). A watchdog alarm or a RAM parity error in the servo module occurred. Replace the servo control module on the main CPU board.
921	SERVO ALARM (3/4 AXIS)	Servo alarm (3rd or 4th axis). A watchdog alarm or a RAM parity error in the servo module occurred. Replace the servo control module on the main CPU board.
924	SERVO MODULE SETTING ERROR	The digital servo module is not installed. Check that the servo control module or servo interface module on the main board is mounted securely.
930	CPU INTERRUPUT	CPU error (abnormal interrupt) The main board is faulty.
940	PCB ERROR	PCB ID error. The main board or the memory module may be defective.
945	SERIAL SPINDLE COMMUNICATION ERROR	Communications error occurred in the serial spindle. The optical fiber may be disconnected.
950	PMC SYSTEM ALARM	Fault occurred in the PMC. The PMC control module on the main board or the RAM module may be defective.

Number	Message	Contents
971	NMI OCCURRED IN SLC	An alarm condition occurred in the interface with an I/O unit. For PMC–RA1 or PMC–RA3, check the connection between the PMC control module on the main board and the I/O Unit. Also, check that the power of the I/O Unit is on and that the interface module is operating normally.
972	NMI OCCURRED IN OTHER MODULE	NMI occurred in a board other than the main board. The main board or the back panel may be defective.
973	NON MASK INTERRUPT	NMI occurred for an unknown reason. The printed board of the power unit or the main board may be defective. Or, there may be noise interference.
974	BUS ERROR	Bus error The main board may be defective.

(10) Alarms displayed on spindle servo unit

Alarm No.	Meaning	Description	Remedy
"A" display	Program ROM abnormality (not installed)	Detects that control program is not started (due to program ROM not installed, etc.)	Install normal program ROM
AL01	Motor overheat	Detects motor speed exceeding specified speed excessively.	Check load status. Cool motor then reset alarm.
AL02	Excessive speed deviation	Detects motor speed exceeding specified speed excessively.	Check load status. Reset alarm.
AL03	DC link section fuse blown	Detects that fuse F4 in DC link section is blown (models 30S and 40S).	Check power transistors, and so forth. Replace fuse.
AL04	Input fuse blown. Input power open phase.	Detects blown fuse (F1 to F3), open phase or momentary failure of power (models 30S and 40S).	Replace fuse. Check open phase and power supply refenerative circuit operation.
AL05	Control power supply fuse blown	Detects that control power supply fuse AF2 or AF3 is blown (models 30S and 40S).	Check for control power supply short circuit . Replace fuse.
AL–07	Excessive speed	Detects that motor rotation has exceeded 115% of its rated speed.	Reset alarm.
AL–08	High input voltage	Detects that switch is flipped to 200 VAC when input voltage is 230 VAC or higher (models 30S and 40S).	Flip switch to 230 VAC.
AL–09	Excessive load on main circuit section	Detects abnormal temperature rise of power transistor radiator.	Cool radiator then reset alarm.
AL–10	Low input voltage	Detects drop in input power supply voltage.	Remove cause, then reset alarm.

Alarm No.	Meaning	Description	Remedy
AL–11	Overvoltage in DC link section	Detects abnormally high direct current power supply voltage in power circuit section.	Remove cause, then reset alarm.
AL–12	Overcurrent in DC link section	Detects flow of abnormally large current in direct current section of power circuit	Remove cause, then reset alarm.
AL–13	CPU internal data memory abnormality	Detects abnormality in CPU internal data memory. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–15	Spindle switch/output switch alarm	Detects incorrect switch sequence in spindle switch/output switch operation.	Check sequence.
AL–16	RAM abnormality	Detects abnormality in RAM for external data. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–18	Program ROM sum check error	Detects program ROM data error. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–19	Excessive U phase current detection circuit offset	Detects excessive U phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–20	Excessive V phase current detection circuit offset	Detects excessive V phase current detection circuit offset. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–24	Serial transfer data error	Detects serial transfer data error (such as NC power supply turned off, etc.)	Remove cause, then reset alarm.
AL–25	Serial data transfer stopped	Detects that serial data transfer has stopped.	Remove cause, then reset alarm.
AL–26	Disconnection of speed detection signal for Cs contouring control	Detects abnormality in position coder signal(such as unconnected cable and parameter setting error).	Remove cause, then reset alarm.
AL–27	Position coder signal disconnection	Detects abnormality in position coder signal (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL–28	Disconnection of position detection signal for Cs contouring control	Detects abnormality in position detection signal for Cs contouring control (such as unconnected cable and adjustment error).	Remove cause, then reset alarm.
AL–29	Short–time overload	Detects that overload has been continuously applied for some period of time (such as restraining motor shaft in positioning).	Remove cause, then reset alarm.
AL–30	Input circuit overcurrent	Detects overcurrent flowing in input circuit.	Remove cause, then reset alarm.
AL–31	Speed detection signal disconnection motor restraint alarm or motor is clamped.	Detects that motor cannot rotate at specified speed or it is detected that the motor is clamped. (but rotates at very slow speed or has stopped). (This includes checking of speed detection signal cable.)	Remove cause, then reset alarm.
AL–32	Abnormality in RAM internal to LSI for serial data transfer. This check is made only when power is turned on.	Detects abnormality in RAM interior LSI for serial data transfer. This check is made only when power is turned on.	Remove cause, then reset alarm.
AL–33	Insufficient DC link section charging	Detects insufficient charging of direct current power supply voltage in power circuit section when magnetic contactor in amplifier is turned on (such as open phase and defective charging resistor).	Remove cause, then reset alarm.
AL–34	Parameter data setting beyond allowable range of values	Detects parameter data set beyond allowable range of values.	Set correct data.
AL–35	Excesive gear ratio data setting	Detects gear ratio data set beyond allowable range of values.	Set correct data.

Alarm No.	Meaning	Description	Remedy
AL–36	Error counter over flow	Detects error counter overflow.	Correct cause, then reset alarm.
AL–37	Speed detector parameter setting error	Detects incorrect setting of parameter for number of speed detection pulses.	Set correct data.
AL–39	Alarm for indicating failure in detecting 1–rotation signal for Cs contouring control	Detects 1–rotation signal detection failure in Cs contouring contorl.	Make 1–rotation signal adjustment. Check cable shield status.
AL–40	Alarm for indicating 1–rotation signal for Cs contouring control not detected	Detects that 1–rotation signal has not occurred in Cs contouring control.	Make 1–rotation signal adjustment.
AL–41	Alarm for indicating failure in detecting position coder 1–rotation signal.	Detects failure in detecting position coder 1–rotation signal.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL–42	Alarm for indicating position coder 1–rotation signal not detected	Detects that position coder 1–rotation signal has not issued.	Make 1–rotation signal adjustment for signal conversion circuit.
AL–43	Alarm for indicating disconnection of position coder signal for differential speed mode	Detects that main spindle position coder signal used for differential speed mode is not connected yet (or is disconnected).	Check that main spindle position coder signal is connected to connector CN12.
AL–46	Alarm for indicating failure in detecting position coder 1–rotation signal in thread cutting operation.	Detects failure in detecting position coder 1–rotation signals in thread cutting operation.	Make 1–rotation signal adjustment for signal conversion circuit Check cable shield status.
AL–47	Position coder signal abnormality	Detects incorrect position coder signal count operation.	Make signal adjustment for signal conversion circuit. Check cable shield status.
AL–48	Position coder 1–rotation signal abnormality	Detects that occurrence of position coder 1–rotation signal has stopped.	Make 1–rotation signal adjustment for signal conversion circuit.
AL–49	The converted differential speed is too high.	Detects that speed of other spindle converted to speed of local spindle has exceeded allowable limit in differential mode.	Check the position coder state of the other side.
AL–50	Excessive speed command calculation value in spindle synchronization control	Detects that speed command calculation value exceeded allowable range in spindle synchronization control.	Check parameters such as a position gain.
AL–51	Undervoltage at DC link section	Detects that DC power supply voltage of power circuit has dropped (due to momentary power failure or loose contact of magnetic contactor).	Remove cause, then reset alarm.
AL–52	ITP signal abnormality I	Detects abnormality in synchronization signal (ITP signal)with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL–53	ITP signal abnormality II	Detects abnormality in synchronization signal (ITP signal) with CNC (such as loss of ITP signal).	Remove cause, then reset alarm.
AL–54	Overload current alarm	Detects that excessive current flowed in motor for long time.	Check if overload operation or frequent acceleration/deceleration is performed.
AL–55	Power line abnormality in spindle switching/output switching	Detects that switch request signal does not match power line status check signal.	Check operation of magnetic contractor for power line switching. Check if power line status check signal is processed normally.

H OPERATION OF PORTABLE TAPE READER

Portable tape reader is the device which inputs the NC program and the data on the paper tape to CNC.

- **Names and descriptions of each section**

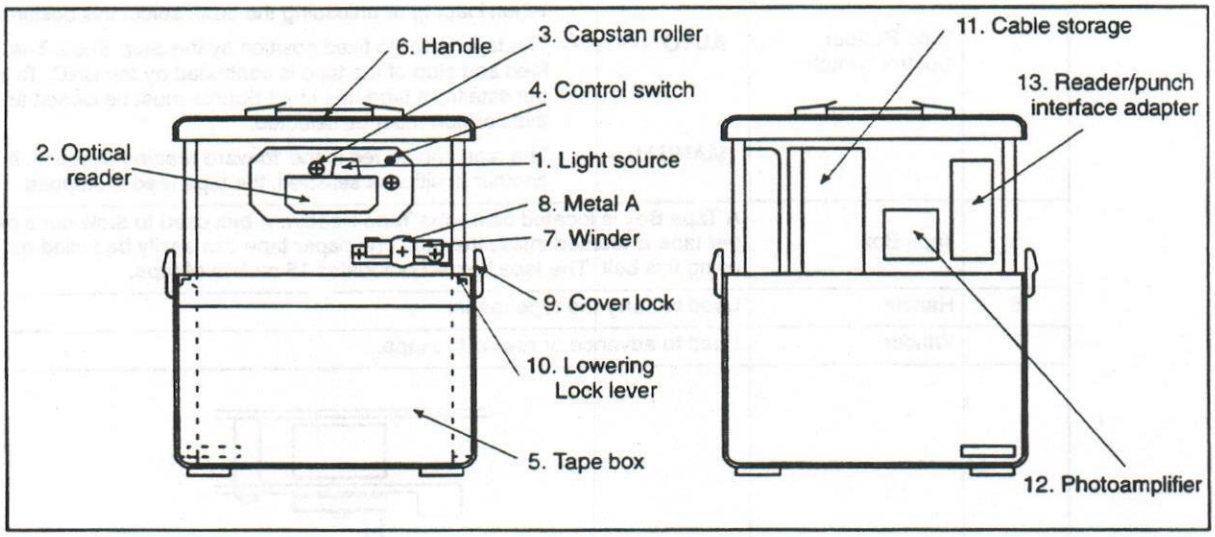
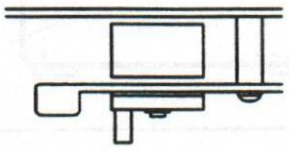
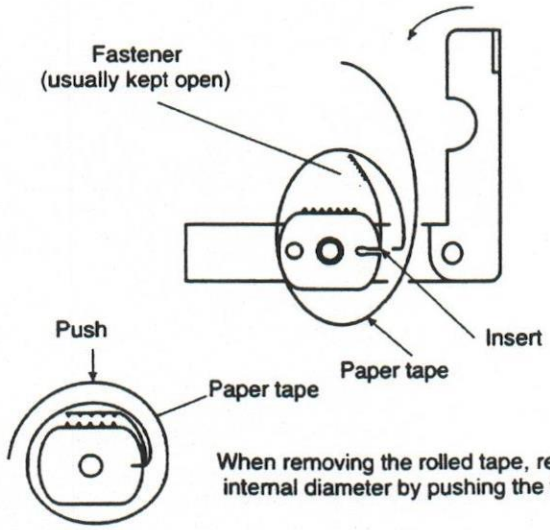


Table 1 Description of Each Section

No.	Name	Descriptions
1	Light Sources	An LED (Light emitting diode) is mounted for each channel and for the feed hole (9 diodes in total). A built–in Stop Shoe functions to decelerate the tape. The light source is attracted to the optical reader by a magnet so that the tape will be held in the correct position. This unit can be opened upward, by turning the tape reader control switch to the RELEASE position (this turns off the magnet).
2	Optical Reader	Reads data punched on the tape, through a glass window. Dust or scratches on the glass window can result in reading errors. Keep this window clean.
3	Capstan Roller	Controls the feeding of tape as specified by the control unit.
4	Tape Reader Control Switch	A 3–position switch used to control the Tape Reader. **RELEASE** ----- The tape is allowed to be free, or used to open the light-source. When loading or unloading the tape, select this position. **AUTO** -------- The tape is set to fixed position by the Stop Shoe. The feed and stop of the tape is controlled by the CNC. To input data from tape, the Light Source must be closed and this position must be selected. **MANUAL** ----- The tape can be fed in the forward reading direction. if another position is selected, the tape feed is stopped.
5	Tape Box	A Tape Box is located below the Tape Reader. A belt used to draw out a paper tape is located inside the box. The paper tape can easily be pulled out using this belt. The tape box accomodates 15 meters of tape.
6	Handle	Used to carry the tape reader.
7	Winder	Used to advance or rewind the tape.
8	Metal A	  When removing the rolled tape, reduce the internal diameter by pushing the fastener.
9	Cover lock	Be sure to use the lock for fastening the cover before carrying the tape reader.

No.	Name	Descriptions
10	Lowering lock lever	When the tape reader is raised, the latch mechanism is activated to fix the tape reader. Thus, the tape reader is not lowered. The latch is locked with the lowering lock lever. The latch is therefore not unlocked even when the tape reader is raised with the handle. When the latch is locked, the lever is horizontal. To store the tape reader in the box, push the lever to release the lock, then raise the tape reader with the handle to unlock the latch. When the latch is unlocked, the tape reader can be stored in the box. When storing the tape reader, secure it with the cover lock.
11	Cable storage	Used to store rolled power and signal cables. The cable length is 1.5 m.
12	Photoamplifier	For the tape reader
13	Reader/punch interface adapter	200 VAC input and 5 VDC output power and reader/punch interface adapter PCB

Procedure for Operating the Portable Tape Reader

Preparations

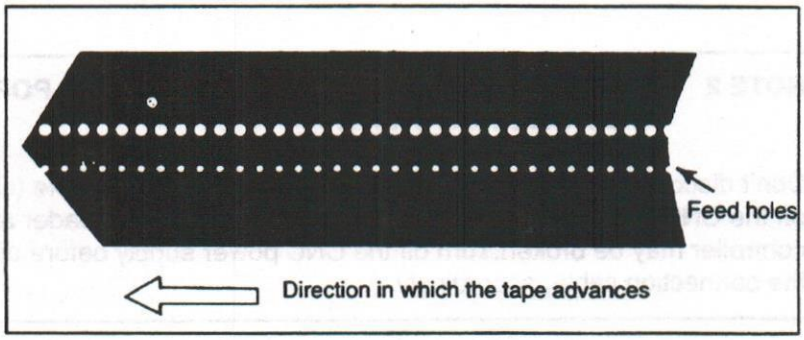
1 Unlock the cover locks **9**. Raise the tape reader with the handle **6** until it clicks, then lower the tape reader. The tape reader then appears and is secured. Check that the lowering lock levers **10** are horizontal.

2 Take out the signal and power cables from the cable storage **11** and connect the signal cable with the CNC reader/punch interface port and the power cable with the power supply.

Setting the tape

3 Turn the control switch to the RELEASE position.

4 Lift the Light Source Unit, and insert an NC tape between the gap. The tape must be positioned as shown in the figure, when viewed looking downward.



5 Pull the tape until the top of the tape goes past the Capstan roller.

6 Check that the NC tape is correctly positioned by the Tape Guide.

7 Lower the Light Source.

8 Turn the switch to the AUTO position.

9 Suspend the top and rear–end of the tape in the Tape Box.

Removing the tape

10 Turn the switch to the RELEASE position.

11 Lift the Light Source and remove the tape.

12 Lower the Light Source

Storage

13 Store the cables in the cable storage **11**.

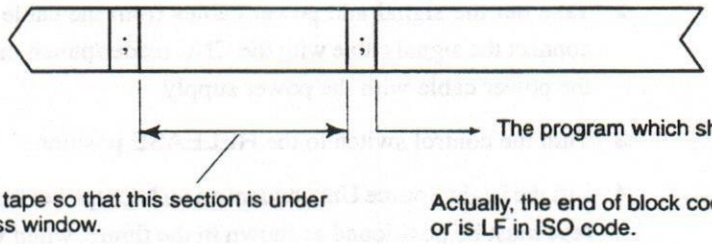
14 Push the lowering lock lever **10** at both sides down.

15 Raise the tape reader with the handle **6** to unlock the latch, then gently lower it.

16 Lock the cover lock **9** and carry the tape reader with the handle **6**.

NOTE 1 SETTING OF A TAPE

When the NC tape is loaded, the Label Skip function activates to read but skip data until first End of Block code (CR in EIA code or LF in ISO code) is read. When loading an NC tape, the location within the tape, from which data reading should be started must properly be selected and the NC tape should be set as shown in the figure below.


;
;
The program which should be read
Set the tape so that this section is under the glass window.
Actually, the end of block code (;) is CR in EIA code or is LF in ISO code.

NOTE 2 DISCONNECTION AND CONNECTION OF A PORTABLE TAPE READER CONNECTION CABLE

Don't disconnect or connect CNC tape reader connection cable (signal cable) without turning off the CNC power supply, otherwise the PCB of the tape reader and master PCB of CNC controller may be broken.Turn off the CNC power supply before disconnecting or connecting the connection cable, accordingly.

[Numbers]

2nd reference position return (G30), 66

[A]

A block without tool movement, 151

A block without tool movement specified at start–up, 139

A block without tool movement specified in offset mode, 151

A block without tool movement specified together with offset cancel, 154

A command to the MPG exceeding rapid traverse rate, 271

Abbreviations of arithmetic and logic operation commands, 181

Abbreviations of custom macro word, 363

About this manual, 3

ABSOLUTE AND INCREMENTAL PROGRAMMING (G90, G91), 74

Absolute commands, 20

Acceleration/deceleration for rapid traverse, 268

Actual Feedrate Display, 417

Actual feedrate display of feed per revolution, 418

Actual feedrate display on the other screen, 418

Actual feedrate value, 417

Alarm, 307

ALARM AND SELF–DIAGNOSIS FUNCTIONS, 311

ALARM DETAIL screen, 455

ALARM DISPLAY, 228, 312

ALARM HISTORY DISPLAY, 314

ALARM LIST, 489

Alarm screen, 312

Alarm status, 453

Alarms during background editing, 365

Altering a Word, 354

An alarm while a program is output, 326

Angle units, 180

Another method for alarm displays, 312

Applicable models, 3

Arc radius, 44

Argument specification, 190, 196, 197, 198

ARITHMETIC AND LOGIC OPERATION, 180

ATAN function, 180

AUTOMATIC ACCELERATION/DECELERATION, 54, 60

Automatic Coordinate System Setting, 71

AUTOMATIC INSERTION OF SEQUENCE NUMBERS, 398

AUTOMATIC OPERATION, 219, 277

Automatic operation control, 175

Automatic operation status, 452

Automatically setting the coordinate system, 266

AUXILIARY FUNCTION, 86

AUXILIARY FUNCTION (M FUNCTION), 87

Auxiliary Function Lock, 222, 297

Availability of manual pulse generator in Jog mode (JHD), 271

Availability of manual pulse generator in TEACH IN JOG mode (THD), 271

Available thread cutting commands, 50

Axis moving status/dwell status, 452

Axis No. (P_), 211

[B]

BACKGROUND EDITING, 364

Battery for Absolute pulse coder , 461

Battery for Memory Backup, 461

Block, 27

Block configuration (word and address), 96

Block containing G40 and I_J_K_, 155

Bracket nesting, 182

Brackets, 182

BRANCH AND REPETITION, 185

Buffering the next block in other than tool nose radius compensation mode (G41, G42) (normally prereading one block), 202

Buffering the next block in tool nose radius compensation mode (G41, G42), 203

[C]

Call, 189, 194, 199

Call nesting, 191, 194

Cancel mode, 136

Cancellation, 194, 294

CANNED CYCLE (G90, G92, G94), 105

CHAMFERING AND CORNER R, 114

Change in the offset direction in the offset mode, 146

Change of modes, 268

Changing of Tool Offset Value (Programmable Data Input) (G10), 168

Changing offset values during automatic operation, 433

Changing parameter PASSWD, 367

Changing the offset value, 166

CHARACTER–TO–CODES CORRESPONDENCE TABLE, 488

Check by Running the Machine, 221

CHECKING BY SELF–DIAGNOSTIC SCREEN, 315

Checking contents of the memory, 402

Checking during search, 360

Checkpoint for the forbidden area, 309

CIRCULAR INTERPOLATION (G02,G03), 42

Clamp of maximum spindle speed, 80

Close command PCLOS, 208

Collation, 324

Command cancelling the offset vector temporality, 150

COMMAND FOR MACHINE OPERATIONS – MISCELLANEOUS FUNCTION, 25

Command of circular interpolation X, Z, 45
Command of offset value, 127
Comment, 374
Comment section, 92
Common variable, 335
Common variables, 334
COMPENSATION FUNCTION, 117
Compensation values for a program created in diameter programming, 435
Concept of word and editing unit, 349
Conditional Branch (IF Statement), 185
Conditional expression, 185
Configuration of the Help Screen, 458
CONSTANT LEAD THREADING (G32), 46
CONSTANT SURFACE SPEED CONTROL (G96, G97), 80
Constant surface speed control cancel command, 80
Constant surface speed control command, 80
Constant surface speed control command (G96), 81
Constant surface speed control for rapid traverse (G00), 83
Constant surface speed control for threading, 82
Constant values that can be used in <expression>, 205
CONTROLLED AXES, 31, 32
Coordinate display, 416
COORDINATE SYSTEM, 17, 67
Coordinate System on Part Drawing and Coordinate System Specified by CNC – Coordinate System, 17
COORDINATE VALUE AND DIMENSION, 73
Copying a Program, 344
Corner movement, 152
Correction in Chamfering and Corner Arcs, 163
Correction of interference in advance, 158
Correspondence between parameter numbers and program numbers, 196, 197, 198
Counter Input of Offset Value, 436
CREATING A PROGRAM IN THE CNC LANGUAGE ON THE MDI PANEL, 396
CREATING A PROGRAM WITH GUIDANCE PROGRAMMING, 369
CREATING PROGRAMS, 368
CREATING PROGRAMS IN TEACH IN MODE, 400
Criteria for detecting interference, 156
CRT/MDI PANELS AND LCD/MDI PANELS, 232
Current Block Display Screen, 423
Current mode, 452
Current position , 178
Current Position Display, 228, 425
Current time, 453
CUSTOM MACRO, 169
CUTTING FEED, 56
Cutting feedrate clamp, 58
CUTTING SPEED – SPINDLE SPEED FUNCTION, 23
CUTTING TIME, 442
CYCLE TIME, 419, 442

[D]

DATA and TIME, 442
DATA INPUT/OUTPUT, 230, 318
Data output command BPRNT, 206
Data output command DPRNT, 207
Data protection key, 226, 403
Decimal point input, 432
DECIMAL POINT PROGRAMMING, 76
Deleting a Block, 341, 356
Deleting a Program, 343
Deleting a Word, 355
Deleting All Programs, 361
DELETING BLOCKS, 356
Deleting More Than One Program by Specifying a Range, 362
Deleting Multiple Blocks, 357
Deleting One Program, 361
DELETING PROGRAMS, 361
Description of each display, 452
Detailed information of alarms, 454
Details of memory used, 428
DETAILS OF TOOL NOSE RADIUS COMPENSATION, 136
DIAMETER AND RADIUS PROGRAMMING, 78
Diameter or radius value, 438
Diameter programming/radius programming, 21
Differences between macro calls and subprogram calls, 189
Differences from NC statements, 184
Direct Input of Tool Offset Value, 434
Direction of Imaginary Tool Nose, 125
Direction of the circular interpolation, 43
Disabling entry of compensation values, 433
DISPLAY, 227
Display including compensation values, 413, 415
Display of Run Time and Parts Count, 419
Display of software configuration, 263
Display of the PROGRAM RESTART page, 205
Display of the servo axes, 420
Display of the spindle axes, 420
Display on the other screen, 419
Display procedure for the actual feedrate on the current position display screen, 417
Display procedure for the current position screen in the workpiece coordinate system, 413
Display procedure for the current position screen with the relative coordinate system, 414
Displaying and Entering Setting Data, 439
Displaying and Setting Custom Macro Common Variables, 443
DISPLAYING AND SETTING DATA, 224
Displaying and setting operator's setting data, 225
Displaying and Setting Parameters, 226, 447

Displaying and Setting Pitch Error Compensation Data, 449
Displaying and Setting Run Time,Parts Count, and Time, 441
Displaying and Setting the Software Operator's Panel, 444
Displaying and Setting the Tool Offset Value, 431
Displaying during automatic operation, 425
Displaying Memory Used and a List of Programs, 427
Displaying radius and TIP, 433
Displaying the Contents of a Program, 338
Displaying the Program Number and Sequence Number, 451
DISPLAYING THE PROGRAM NUMBER, SEQUENCE NUMBER, AND STATUS, AND WARNING MESSAGES FOR DATA SETTING OR INPUT/OUTPUT OPERATION, 451
Displaying the Status and Warning for Data Setting or Input/Output Operation, 452
Displaying variable values, 172
Distance from the start point to the center of arc, 43
Distance moved on an arc, 43
Distance to go, 416
Divisor, 183
DNC OPERATION, 218, 288
DRY RUN, 221, 301
Dry run feedrate, 302
DWELL (G04), 59

[E]

EDITING A PART PROGRAM, 223
Editing a program during a guidance programming MDI operation, 287
Editing in CNC Language, 347
EDITING OF CUSTOM MACROS, 363
EDITING PROGRAMS, 336
Editing unit, 363
EDITING WITH GUIDANCE PROGRAMMING, 338
Effective time for a forbidden area, 309
Efficient use of memory, 327
EMERGENCY STOP, 306
Emergency stop or reset status, 452
Enabling and disabling manual intervention and return, 294
End Face Turning Cycle (G94), 109
End position for the arc is not on the arc, 144
Erasing the program, 286
Error codes, 313
Example of changing T15 to M15, 354
Example of deleting a block of No.01234, 356
Example of deleting blocks from a block containing N01234 to a block containing N56789, 357
Example of deleting X100.0, 355
Example of Inserting T15, 353
Explanation of the keyboard, 233
EXTERNAL I/O DEVICES, 256
EXTERNAL OUTPUT COMMANDS, 206

[F]

Face cutting cycle, 109
Face cutting cycle (G94), 113
Face taper cutting cycle (G94), 113
FANUC FA Card, 259
FANUC Floppy Cassette, 258
FANUC Handy File, 258
FANUC PPR, 259
FEED FUNCTIONS, 53, 54
Feed hold, 205, 280
Feed per minute (G98), 57
Feed per revolution (G99), 57
FEED–FEED FUNCTION, 15
Feedrate, 44
FEEDRATE OVERRIDE, 221, 299
FILE DELETION, 323
File number after the file is deleted, 323
File output location, 326
FILE SEARCH, 321
File search by N–9999, 321
FILES, 319
First return to the reference position after the power has been turned on (without an absolute position detector), 66
FREE PURPOSE, 442
Full–circle programming, 43
Function Keys, 236
FUNCTION KEYS AND SOFT KEYS, 235
Function Selection Soft Keys, 255
FUNCTIONS TO SIMPLIFY PROGRAMMING, 104

[G]

G code system A, 74
G code system B or C, 74
G40, G41, G42, 136
GENERAL FLOW OF OPERATION OF CNC MACHINE TOOL, 5
General Precautions for Offset Operations, 166
General purpose switches, 445
General Screen Operations, 235
Geometry offset and wear offset, 436
Guidance Programming, 375

[H]

Handle interruption, 220
Handle/incremental feed mode, 269

Heading a Program, 352

HELP FUNCTION, 454

How to Indicate Command Dimensions for Moving the Tool – Absolute, Incremental Commands, 20

How to Use Canned Cycles (G90, G92, G94), 112

How to use nomograph, 479

How to View the Position Display Change without Running the Machine, 222

[I]

I/O CHANNEL, 440

Imaginary Tool Nose, 123

Imaginary tool nose direction, 127

INCH/METRIC CONVERSION (G20, G21), 75

INCORRECT THREADED LENGTH, 478

INCREMENT SYSTEM, 33

Incremental commands, 21

INCREMENTAL FEED, 269

Incrementing the number of machined parts, 419

Infinite loops, 187

Inner side and outer side, 137

Input Command from MDI, 165

INPUT UNIT, 440

Inputting a Program, 324

INPUTTING AND OUTPUTTING PARAMETERS AND PITCH ERROR COMPENSATION DATA, 330

Inputting Custom Macro Common Variables, 334

Inputting multiple programs from an NC tape, 325

Inputting Offset Data, 328

Inputting Parameters, 330

Inputting Pitch Error Compensation Data, 332

INPUTTING/OUTPUTTING CUSTOM MACRO COMMON VARIABLES, 334

Inserting a Block, 342

Inserting a Word, 353

INSERTING, ALTERING AND DELETING A WORD, 349

Interface signals, 174

Interference Check, 156

INTERPOLATION FUNCTIONS, 38

Interruption of manual operation, 152

ISO code, 327

[J]

JOG FEED, 267

Jog feed and arrow keys, 445

[K]

Key Input and Input Buffer, 253

[L]

Leader section, 91

Lighting the lamp when the programmed position does not coincide with the reference position, 66

Linear Acceleration/Deceleration after Interpolation for Cutting Feed, 62

Linear interpolation, 41

LINEAR INTERPOLATION (G01), 41

LIST OF FUNCTIONS AND TAPE FORMAT, 472

Load meter, 420

Local variable levels, 191

Lower digit of T code specifies geometry and wear offset number, 119

Lower digit of T code specifies wear offset number and higher digit number specifies tool selection number and geometry offset number, 119

[M]

M code, 198

M codes not locked by auxiliary function lock, 298

M, S, T command by only machine lock, 298

M00 (Program stop), 87

M01 (Optional stop), 87

M02, M03 (End of program, 87

M98 (Calling of sub–program), 87

M99 (End of subprogram), 87

MACHINE COORDINATE SYSTEM, 68, 416

Machine Lock, 222, 297

MACHINE LOCK AND AUXILIARY FUNCTION LOCK, 297

Machine lock and mirror image, 294

Machining a groove smaller than the tool nose radius, 161

Machining a step smaller than the tool nose radius, 162

Machining an inside corner at a radius smaller than the tool nose radius, 161

Machining using the end of cutter – Tool length compensation function (See II–15.1), 29

Macro alarms, 175

MACRO CALL, 189, 287

Macro call and single block, 304

Macro Call Using an M Code, 197

Macro Call Using G Code, 196

MACRO STATEMENTS AND NC STATEMENTS , 184

Main program and subprogram, 28, 89

Major addresses and ranges of command values, 97

MANUAL ABSOLUTE ON AND OFF, 272

Manual absolute on/off, 294

MANUAL HANDLE FEED, 270

MANUAL HANDLE INTERRUPTION, 289
MANUAL INTERVENTION AND RETURN, 294
MANUAL OPERATION, 216, 264
Manual operation after a feed hold, 273
Manual operation after the end of block, 273
Manual operation during tool nose radius compensation, 274
MANUAL REFERENCE POSITION RETURN, 66, 216, 265
Manual synchronous feed, 268
MAXIMUM STROKES, 34
MDI mode, 294
MDI operation, 205, 218, 426
MDI OPERATION (GUIDANCE PROGRAMMING MDI OPERATION), 281
Meaning of symbols, 137
Memory area, 287
MEMORY OPERATION, 218, 278, 279
METHOD OF REPLACING BATTERY, 461
Methods of setting the two coordinate systems in the same position, 18
MIRROR IMAGE, 177, 292, 440
Mixture of argument specifications I and II, 190
Modal Call (G66), 194
Modal call nesting, 194
Modal G codes, 425
Modal information, 178, 426
Modifying the Contents of a Program, 340
Movement direction of an axis to the rotation of MPG (HNG), 271
Moving the tool again, 266
MULTIPLE M COMMANDS IN A SINGLE BLOCK, 88
MULTIPLE-THREAD CUTTING, 50

[N]

NAMES OF AXES, 32
Nesting, 187
Nesting of calls using G codes, 196
Next Block Display Screen, 424
NOMOGRAPHS, 477
Notes on diameter programming/radius programming for each command, 78
NOTES ON READING THIS MANUAL, 7
Notes on Tool Nose Radius Compensation, 133
Number of lines in a program, 287
Number of machined parts, 177
Number of MPGs, 271
Number of tool compensation, 167

[O]

Offset, 120, 294
Offset cancel, 131
OFFSET DATA INPUT AND OUTPUT, 328
Offset Number, 119
Offset Number and Offset Value, 126
Offset value, 224
Offset vector, 120
Omission of the decimal point, 171
On the memo record, 327
Open command POPEN, 206
Operating Monitor Display, 420
OPERATING TIME, 442
Operation check, 200
Operation during Search, 360
Operation error, 182
Operation in EDIT mode, 205
Operation method, 454
OPERATION METHOD screen, 456
OPERATION OF PORTABLE TAPE READER, 503
OPERATIONAL DEVICES, 231
Operators, 186
Optional block skip, 98, 205, 280
Optional stop (M01), 280
Order in which programs are displayed in the program library list, 428
Order in which programs are registered, 429
Other method, 432
Outer Diameter/Internal Diameter Cutting Cycle (G90), 105
Output format, 329, 331, 333, 335
Outputting a Program, 326
Outputting a program after file heading, 326
Outputting Custom Macro Common Variable, 335
Outputting Offset Data, 329
Outputting Parameters, 331
Outputting Pitch Error Compensation Data, 333
Overall Position Display, 416
Overcutting by Tool Nose Radius Compensation, 161
Override, 54, 294
Override during thread, 299
Override Range, 299
Overrun amount of stored stroke limit, 310
OVERTRAVEL, 307
Overtravel during automatic operation, 307
Overtravel during manual operation, 307
OVERVIEW OF TOOL NOSE RADIUS COMPENSATION, 123

[P]

Parameter list, 448
Parameter setting, 200, 419
Parameter setting value (R_), 211

Parameter table, 454
PARAMETER TABLE screen, 457
PARAMETER WRITE, 440
Parameters that require turning off the power, 448
PART COUNT, 419
PART DRAWING AND TOOL MOVEMENT, 16
PARTS COUNT, 442
Parts Count Display, Run Time Display, 229
PARTS REQUIRED, 441
PARTS TOTAL, 441
PASSWORD FUNCTION, 366
Pitch error compensation, 332
Portable Tape Reader, 260
Position display, 290
Position Display in the Relative Coordinate System, 414
Position Display in the Workpiece Coordinate System, 413
Position of the decimal point, 190
POSITIONING (G00), 39
Power Disconnection, 263
POWER ON, 442
POWER ON/OFF, 261
PREPARATORY FUNCTION (G FUNCTION), 35
Presetting by setting a coordinate system, 415
Priority of operations, 181
Procedure for altering a word, 354
Procedure for background editing, 364
Procedure for counter input of offset value, 436
Procedure for deleting a block, 356
Procedure for deleting a word, 355
Procedure for deleting all programs, 361
Procedure for deleting more than one program by specifying a range, 362
Procedure for deleting multiple blocks, 357
Procedure for deleting one program, 361
Procedure for direct input of tool offset value, 434
Procedure for displaying and setting custom macro common variables, 443
Procedure for displaying and setting parameters, 447
Procedure for Displaying and Setting Run Time, Parts Count and Time, 441
Procedure for displaying and setting the pitch error compensation data, 450
Procedure for displaying and setting the software operator's panel, 444
Procedure for displaying memory used and a list of programs, 427
Procedure for displaying overall position display screen, 416
Procedure for displaying run time and parts count on the current position display screen, 419
Procedure for displaying the current block display screen, 423
Procedure for displaying the next block display screen, 424
Procedure for displaying the operating monitor, 420
Procedure for displaying the program check screen, 425
Procedure for displaying the program contents, 422
Procedure for displaying the program screen for MDI operation, 426
Procedure for displaying variable values, 173
Procedure for enabling/displaying parameter writing , 448
Procedure for Heading a Program, 352
Procedure For Incremental Feed, 269
Procedure for inserting a word, 353
Procedure for inserting, altering and deleting a word, 349
Procedure for Jog Feed, 267
Procedure for Manual Handle Feed, 270
Procedure for Manual Reference Position Return, 265
Procedure for Operating the Portable Tape Reader, 505
Procedure for Poser Disconnection, 263
Procedure for program number search, 358
Procedure for replacing batteries for absolute pulse coder, 464, 465
Procedure for replacing battery for memory back–up, 462
Procedure for scanning a program, 350
Procedure for searching a word (valid for only MDI mode), 351
Procedure for searching an address (valid for only MDI mode), 351
Procedure for sequence number search, 359
Procedure for setting and displaying the tool offset value and the tool nose radius compensation value, 431
Procedure for setting the setting data, 439
Procedure for setting the workpiece coordinate system shifting amount, 437
Procedure of turning on the power, 261
Procedure to reset all axes, 415
Procedure to set the axis coordinate to a specified value, 415
PROCESSING MACRO STATEMENTS, 202
Processing time, 187
Program, 27
Program Check Screen, 425
Program components, 90
PROGRAM COMPONENTS OTHER THAN PROGRAM SECTIONS, 91
PROGRAM CONFIGURATION, 26, 89
Program Contents Display, 422
Program Display, 227, 425
Program editing status, 453
Program end, 99
Program end (M02, M30), 280
PROGRAM INPUT/OUTPUT, 324
Program library list, 428
Program Menu Screen, 369
Program name, 428
Program number, 94
PROGRAM NUMBER SEARCH, 358
Program registration, 286
Program registration in the background, 325
Program Screen for MDI Operation, 426

PROGRAM SECTION CONFIGURATION, 94
Program section configuration, 90
Program selection, 219
Program start, 92
Program stop (M00), 280
PROGRAMMABLE PARAMETER ENTRY (G10), 210
Protect switch, 320, 323
PUNCH CODE, 440
Punching all programs, 327
Punching programs in the background, 327

[R]

RADIUS DIRECTION ERROR AT CIRCLE CUTTING, 485
RANGE OF COMMAND VALUE, 475
Range of variable values, 171
RAPID TRAVERSE, 55
RAPID TRAVERSE OVERRIDE, 300
Rapid traverse prior to reference position return, 268
Re–locking, 367
REFERENCE POSITION, 64
Reference position, 64
Reference Position (Machine–Specific Position), 16
Reference position return, 65
Reference position return (G28), 66
Reference position return and single block, 304
Reference position return check, 65
Reference position return check (G27), 66
Reference position return check in an offset mode, 66
Reference position return completion LED, 266
Reference position return under Machine Lock, 298
Referencing variables, 171
Registering a position with compensation, 402
Registering a Program, 371
Registering a program having the same number as that of a previously registered program, 325
Registering commands other than position commands, 402
REGISTERING CUSTOM MACRO PROGRAMS, 204
Related manuals, 4
Relation with other functions, 290
Releasing overtravel, 307
Releasing the alarms, 309
Removing the tape, 506
Renaming a Program, 345
Repetition, 196, 197, 198
Repetition (WHILE Statement), 186
REPLACING BATTERIES FOR ABSOLUTE PULSE CODER, 464
REPLACING BATTERIES FOR ABSOLUTE PULSE CODER (SERVO AMPLIFIER CONVERTER UNIT ONLY FOR SERIES 20 OR α SERIES SERVO AMPLIFIER MODULE), 465
REPLACING CNC BATTERY FOR MEMORY BACK–UP, 462
Request for floppy replacement, 319
Required setting, 209
Reset, 280
Reset of the alarm, 313
Resetting relative coordinates, 416
Restart, 286
Retracting along two axes, 435
Return operation, 294
ROUND function, 180
Rounding up and down to an integer, 181
RUN TIME, 419
Run time and part count settings, 442

[S]

SAFETY FUNCTIONS, 305
Screen Displayed at Power–on, 262
Screen indicating module setting status, 262
Screen transition chart, 403
SCREENS DISPLAYED BY <POS> FUNCTION KEY, 412
SCREENS DISPLAYED <OFFSET SETTING> BY FUNCTION KEY, 430
SCREENS DISPLAYED BY FUNCTION KEY <SYSTEM>, 446
SCREENS DISPLAYED BY FUNCTION KEY <prog> (IN MEMORY MODE OR MDI MODE), 421
SCREENS DISPLAYED BY FUNCTION KEY <PROG> (IN THE EDIT MODE), 427
Screens on which jog feed is valid, 445
Searching in sub–program, 360
SELECTION OF TOOL USED FOR VARIOUS MACHINING – TOOL FUNCTION, 24
Sequence number and block, 95
SEQUENCE NUMBER SEARCH, 205, 359
SEQUENCE STOP, 440
Setting 0 in parameter PASSWD, 367
Setting a Workpiece Coordinate System (Method Using G50), 69
SETTING AND DISPLAYING DATA, 403
Setting data, 448
Setting of X axis offset value, 435
Setting of Z axis offset value, 434
Setting parameter PASSWD, 367
Setting parameters with external input/output devices, 448
Setting range of offset value, 127
Setting the forbidden area for the 16–TB (two–path control), 309
Setting the relative coordinates, 414
Setting the tape, 505

Setting the workpiece coordinate system for constant surface speed control, 81

Setting the Workpiece Coordinate System Shifting Amount, 437

Shift values and coordinate system setting, 438

Shift values and coordinate system setting command, 438

Signs of numbers specified in the taper cutting cycle, 106, 110

SIMPLE CALCULATION OF INCORRECT THREAD LENGTH, 480

Simple Call (G65), 189

SINGLE BLOCK, 205, 222, 294, 303

Single block during a canned cycle, 304

SKIP FUNCTION (G31), 51

Slot status display, 262

Soft Keys, 237

Specifiable start angle range, 50

Specification of G41/G42 in G41/G42 mode, 131

Specifying G28 (automatic return to the reference position) in the offset mode, 149

Specifying the sequence number for the return destination in the main program, 102

SPECIFYING THE SPINDLE SPEED VALUE DIRECTLY (S5–DIGIT COMMAND), 80

SPECIFYING THE SPINDLE SPEED WITH A CODE, 80

Speed meter, 420

SPINDLE SPEED FUNCTION, 79

Start and stop, 219

Start angle, 50

Start angle increment, 50

Start–up, 130, 137

State in which an auxiliary function is being executed, 452

Status the machine lock being turned on, 66

STATUS WHEN TURNING POWER ON, WHEN CLEAR AND WHEN RESET, 486

Stopping and terminating memory operation, 280

Stopping the punch, 327

Stored stroke limit 1, 308

Straight cutting cycle, 105

Straight cutting cycle (G90), 112

STROKE CHECK, 308

SUBPROGRAM, 100

Subprogram call, 100

Subprogram Call Using an M Code, 198

Subprogram Calls Using a T Code, 199

Subprogram configuration, 100

Subprogram nesting, 287

Surface speed specified in the G96 mode, 82

SYSTEM VARIABLES, 174

[T]

T Code for Tool Offset, 119

Tangential speed constant control, 56

TAPE CODE LIST, 469

Tape end, 93

Tape start, 91

Taper cutting cycle, 106

Taper cutting cycle (G90), 112

Taper face cutting cycle, 110

Taper thread cutting cycle, 108

Temporary tool nose radius compensation cancel, 149

TEST OPERATION, 296

TESTING A PROGRAM, 221

The center of the arc is identical with the start position or the end position, 145

The distance to return to reference position, 266

The next block to G31 is an absolute command for 1 axis, 52

The next block to G31 is an absolute command for 2 axes, 52

The next block to G31 is an incremental command, 52

The polarity of the offset amount and the tool nose center path, 166

The previous block contains G41 or G42, 155

The tool movement by manual operation, 217

There is no inner intersection, 145

Thread cutting, 14

Thread Cutting Cycle (G92), 107

Time information, 175

Time settings, 442

Tool Compensation and Number of Tool Compensation, 167

Tool compensation values, 174

TOOL COMPENSATION VALUES, NUMBER OF COMPENSATION VALUES, AND ENTERING VALUES FROM THE PROGRAM (G10), 167

TOOL FIGURE AND TOOL MOTION BY PROGRAM, 29

TOOL FUNCTION (T FUNCTION), 84

Tool geometry offset, 121

Tool Geometry Offset and Tool Wear Offset, 118

Tool geometry offset cancel, 121

Tool geometry offset value and tool wear offset value, 435

Tool movement along a straight line, 12

Tool movement along an arc, 13

TOOL MOVEMENT ALONG WORKPIECE PARTS FIGURE–INTERPOLATION, 12

Tool movement around an inner side of a corner ($180^\circ \leqq \alpha$), 138

Tool movement around an inside corner ($180^\circ \leqq \alpha$), 153

Tool movement around an outside corner at an acute angle ($\alpha<90^\circ$), 154

Tool movement around an outside corner at an obtuse angle ($90^\circ \leqq \alpha<180^\circ$), 153

Tool movement around the inside ($\alpha<1^\circ$) with an abnormally long vector, linear → linear, 141

Tool movement around the inside of a corner ($180^\circ \leqq \alpha$), 140

Tool movement around the outside corner at an acute angle ($\alpha<90^\circ$), 143

Tool movement around the outside corner at an obtuse angle ($90^\circ \leqq \alpha<180^\circ$), 142

Tool movement around the outside linear→linear at an acute angle less than 1 degree ($\alpha<1^\circ$), 154

Tool movement around the outside linear→linear at an acute angle less than 1 degree (α<1°), 139

Tool movement around the outside of a corner at an obtuse angle (90° ≦ α<180°), 138

Tool movement around the outside of an acute angle (α<90°), 139

TOOL MOVEMENT BY PROGRAMING – AUTOMATIC OPERATION, 218

Tool movement from point P to point Q (diameter programming is used for the X–axis), 74

Tool Movement in Offset Mode, 140

Tool Movement in Offset Mode Cancel, 153

Tool Movement in Start–up, 138

TOOL MOVEMENT RANGE – STROKE, 30

Tool movement when the moving direction of the tool in a block which includes a G40 command is different from the direction of the workpiece, 131

Tool movement when the workpiece position changes, 130

Tool movement when the workpiece position does not change, 129

Tool movement when two or more blocks without a move command should not be programmed consecutively, 133

Tool nose radius center path with an intersection , 147

Tool nose radius center path without an intersection, 148

Tool nose radius compensation, 127

Tool nose radius compensation G code in the offset mode, 149

Tool nose radius compensation vector, 136

Tool nose radius compensation when a corner arc is inserted, 135

Tool nose radius compensation when chamfering is performed, 135

Tool nose radius compensation when the block is specified from the MDI, 135

Tool nose radius compensation with G90 or G94, 134

Tool nose radius compensation with G92, 134

TOOL OFFSET, 118

Tool offset memory, 432

TOOL PATH AT CORNER, 482

Tool path in a cutting feed, 61

TOOL SELECTION, 85, 119

Tool wear offset, 120

Tool wear offset cancel, 120

Tool wear offset when only a T code is specified, 121

Travel distance display, 290

Travel distance specified with a diameter, 269

Turning on the Power, 261

TV CHECK, 327, 440

TV check (Vertical parity check along tape), 95

Types of variables, 170

[U]

Unconditional Branch (GOTO Statement), 185

Undefined variable, 171, 187

Unit of graph, 420

Using a subprogram only, 103

Using M99 in the main program, 102

[V]

Valid operations, 445

Valid range of tool compensation values, 167

Variable representation, 170

Variable value setting, 200

VARIABLES, 170

[W]

Warning for data setting or input/output operation, 453

Warning Messages, 254

Warning Messages in Guidance Programming, 395

What is a File, 319

When a movement command in the next block is only one axis, 274

When interference is assumed although actual interference does not occur, 160

When it is exceptional, 144

When machining area remains or an alarm is generated, 163

When reset after a manual operation following a feed hold, 274

When shift values become valid, 438

When the next block involves no movement in tool nose radius compensation C (G41, G42) mode, 203

When the next block is not buffered (M codes that are not buffered, G31, etc.), 202

When the next move block is an incremental, 274

Word Search, 350

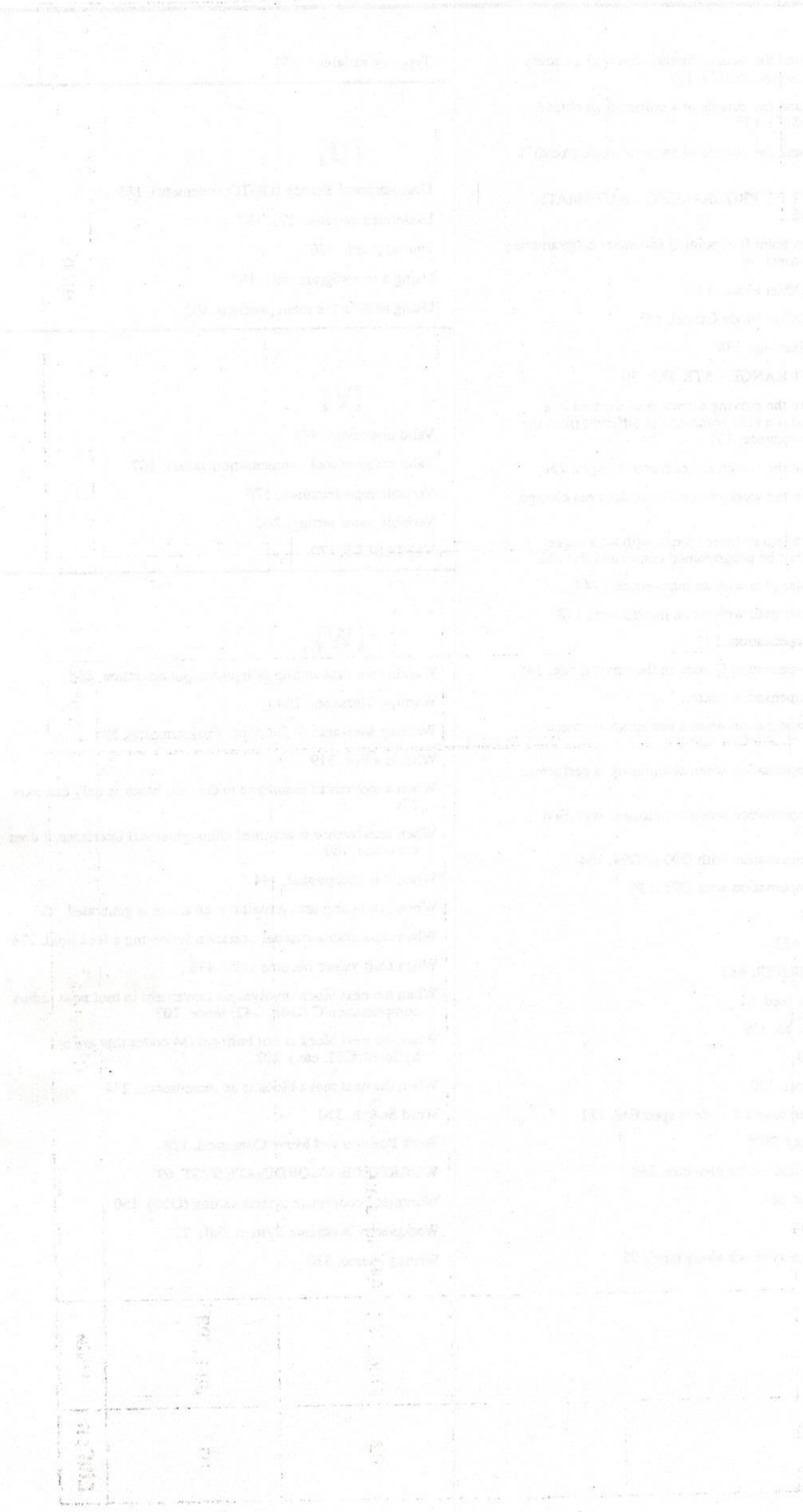
Work Position and Move Command, 128

WORKPIECE COORDINATE SYST, 69

Workpiece coordinate system setting (G50), 150

Workpiece Coordinate System Shift, 72

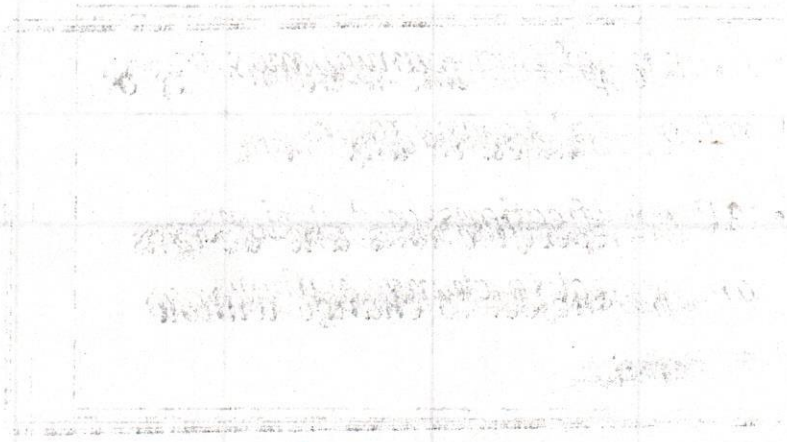
Writing Memo, 320



Revision Record

FANUC Series 20–TA OPERATOR'S MANUAL (B–62204E)

02	Dec., '94	All pages are revised.			
01	Jun., '93	———			
Edition	**Date**	**Contents**	**Edition**	**Date**	**Contents**



- *No part of this manual may be reproduced in any form.*
- *All specifications and designs are subject to change without notice.*

GFZ-62204E/02